Mastercam X2

Advanced Multiaxis Tutorial

Mastercam X2 Advanced Multiaxis Tutorial

September 2006

Mastercam X2

Mastercam® Advanced Multiaxis Tutorial

Date: September 15, 2006
Copyright ©2006 CNC Software, Inc. — All rights reserved.
First Printing: September 15, 2006
Software: Mastercam X2
ISBN: 1-883310-61-X

IMPORTANT NOTICE!

PLEASE READ THIS STATEMENT AND THE SOFTWARE LICENSE AGREEMENT COMPLETELY BEFORE USING THIS SOFTWARE.

BY CONTINUING TO USE THIS SOFTWARE, YOU (EITHER AN INDIVIDUAL OR A SINGLE ENTITY) INDICATE YOUR INTENTION TO BE BOUND BY AND ACCEPT THE TERMS AND CONDITIONS OF THIS SOFTWARE LICENSE. IF YOU DO NOT AGREE TO THESE TERMS AND CONDITIONS, YOU MAY NOT ACCESS OR OTHERWISE USE THIS SOFTWARE AND WILL IN FACT BE PROHIBITED FROM DOING SO. THIS COMPUTER SOFTWARE MAY BE USED ONLY PURSUANT TO THE TERMS AND CONDITIONS SET FORTH BELOW, AND SOLELY IN CONJUNCTION WITH THE ACCOMPANYING SECURITY MECHANISM (UNLESS OTHERWISE SPECIFIED IN THE "EXCEPTIONS TO SECURITY MECHANISM REQUIREMENTS" SECTION OF SUCH TERMS AND CONDITIONS) WHICH MUST BE PRESENT ON YOUR COMPUTER (OR NETWORK AS APPLICABLE) AT ALL TIMES DURING SUCH USE.

Read This If You Have Received This Software From A Third Party: If you received this Software from someone other than an authorized CNC Software, Inc. reseller, you do NOT have a legal software license. (For example, if you acquired this Software simply by buying a machine or Software from the former owner or from an auction, you do **not** have a valid license.) You should contact CNC Software, Inc. (860-875-5006) in order to obtain a new and valid license. **Use of this Software without a valid software license is unlawful, a violation of the Copyright Act and may expose you to criminal liability under 17 United States Code Sec. 506, including fines and damages.**

Software License

If you have paid a license fee, CNC Software, Inc. ("CNC") a Connecticut corporation with its principal place of business at 671 Old Post Road, Tolland, Connecticut, 06084 hereby grants to you a non-exclusive, non-transferable license (the "License") to use this software program (the "Software") and its accompanying documentation (and, if applicable, to permit your authorized employees to use them), solely in accordance with the terms and conditions of this Software License Agreement . You may use the Software solely for your internal business purposes and solely in conjunction with the accompanying hardware or software device, method, scheme or other security measure provided by CNC which allows a user to access the Software and prevents unauthorized access to the Software (the "Security Mechanism"). The Software, any updates to the Software (including any that you download through the Internet), and the Documentation shall hereinafter collectively be referred to as the "Software" and are all governed by this License.

Restrictions

You may not use the Software without a Security Mechanism provided by CNC or CNC's suppliers. When CNC or CNC's suppliers provide you with a single-user Security Mechanism, the Software may only be used (in executable code form only) on a single computer to which the Security Mechanism is physically attached. In the event CNC or CNC's suppliers provide you with a multiple-user Security Mechanism for use over an internal network (a "Network Security Mechanism"), the Software may be used: (a) in executable code form only; (b) only on end-user computers that are connected to the internal network to which the Network

Security Mechanism is attached; and (c) only by the number of users and accessed by the number of end-user computers for which licenses were purchased and as further allowed by the Network Security Mechanism. You may physically transfer the Software from one computer equipped with a single-user Security Mechanism to another only if the Security Mechanism is included in the transfer and is installed with the new computer.

You shall not: (a) copy (except as provided below), adapt, modify the Software; (b) publish, display, disclose or create a derivative work from the Software or any part thereof; (c) de-compile or translate, disassemble, create or attempt to create, by reverse engineering or otherwise, the source code form of the Software from the executable code of the Software; (d) remove any proprietary notices, labels or marks from the Software; (e) sell, rent, lease, distribute or otherwise transfer or provide all or any part of the Software to any person or entity without the prior written consent of CNC; (f) use the Software to provide outsourcing, service bureau, time sharing or other services to any third party; or (g) sublicense, assign, delegate or otherwise transfer your rights in the Software, under the Software License Agreement or any of the related rights or obligations for any reason without the prior written consent of CNC. You shall not circumvent, bypass, modify, reverse engineer, disassemble, disable, alter, enhance or replicate the function of the Security Mechanism in any manner whatsoever. Any attempt to do so shall result in automatic termination of this License without prejudice to all other legal rights and remedies of CNC.

Copying Restrictions

You may make one (1) copy of the Software for backup or archival purposes, provided that you reproduce all proprietary notices of CNC on any such copy.

Non Transferable

You may not transfer or assign the Software or this Software License Agreement or any rights or obligations hereunder. Any attempt to do so will automatically terminate this License without the need for notice. This termination is without prejudice to all other legal rights and remedies of CNC.

Intellectual Property Rights

The Software is and includes intellectual property of CNC. All associated intellectual property rights, including, without limitation, worldwide patent, trademark, copyright and trade secret rights, are reserved. CNC retains all right, title and interest in and copyrights to the Software, regardless of the form or media in or on which the original or other copies may subsequently exist. This Software License Agreement shall not constitute a sale of the Software and no title or proprietary rights to the Software are transferred to you hereby. You acknowledge that the Software is a unique, confidential and valuable asset of CNC, and CNC shall have the right to seek all equitable and legal redress, which may be available to it for the breach or threatened breach of this Software License Agreement including, without limitation, injunctive relief. Unauthorized copying of the Software or failure to comply with the above restrictions shall result in automatic termination of this License and this Software License Agreement without prejudice to all other legal rights and remedies of CNC.

Confidentiality

You acknowledge that the Software contains proprietary trade secrets of CNC and you hereby agree to maintain the confidentiality of the Software using at least as great a degree of care as you use to maintain the confidentiality of your own most confidential information. You agree to reasonably communicate the terms and conditions of this Software License Agreement to those persons employed by you who come into contact with the Software, and to use reasonable best efforts to ensure their compliance with such terms and conditions, including, without limitation, not knowingly permitting such persons to use any portion of the Software for the purpose of deriving the source code of the Software or defeating the Security Mechanism.

Enforcement Obligations

In the event you become aware that any person or entity in your employ or under your control in a manner not authorized by this Software License Agreement is using the Software, you shall immediately use reasonable best efforts to have such unauthorized

use of the Software immediately cease. You shall promptly notify CNC in writing of any unauthorized use of the Software of which you become aware.

Limited Warranties

CNC WARRANTS THAT THE MEDIA ON WHICH THE SOFTWARE IS DISTRIBUTED WILL BE FREE OF DEFECTS IN MATERIAL OR WORKMANSHIP FOR A PERIOD OF THIRTY (30) DAYS AFTER PURCHASE. THE FOREGOING LIMITED WARRANTY EXCLUDES DEFECTS ARISING OUT OF ACCIDENT, NEGLECT, MISUSE, FAILURE OF ELECTRIC POWER AND CAUSES OTHER THAN ORDINARY AND AUTHORIZED USE. EXCEPT FOR THE FOREGOING LIMITED WARRANTY, THE SOFTWARE IS PROVIDED "AS IS, WITH ALL FAULTS." YOUR SOLE REMEDY AND CNC'S SOLE OBLIGATION HEREUNDER SHALL BE, AT CNC'S SOLE OPTION, REPLACEMENT OF THE DEFECTIVE MEDIA OR REFUND OF THE PURCHASE PRICE OF THE SOFTWARE. ANY USE BY YOU OF THE SOFTWARE IS AT YOUR OWN RISK. THIS LIMITED WARRANTY IS THE ONLY WARRANTY PROVIDED BY CNC REGARDING THE SOFTWARE. TO THE MAXIMUM EXTENT PERMITTED BY LAW, CNC DISCLAIMS ALL OTHER WARRANTIES OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, WITHOUT LIMITATION, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. CNC IS NOT OBLIGATED TO PROVIDE ANY UPDATES TO THE SOFTWARE. SHOULD THE SOFTWARE PROVE DEFECTIVE FOLLOWING ITS PURCHASE, YOU (AND NOT CNC, ITS DISTRIBUTOR, OR RETAILER) ASSUME THE ENTIRE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION AND ANY INCIDENTAL OR CONSEQUENTIAL DAMAGES.

Note on Documentation: While CNC makes every effort to ensure that its Documentation for the Software is accurate and up-to-date, it cannot guarantee the Documentation at all times represents the latest operation and functionality of the Software. The content of the "Getting Started Guide," "User Manual" and other documentation for the Software is provided for informational purposes only. The content of the Documentation may be changed without notice to you. **CNC expressly disclaims any warranty or representation that the Documentation is an accurate and/or current reflection of the Software's operation and performance.**

Limitation of Liability

IN NO EVENT WILL CNC, OR ITS EMPLOYEES, SHAREHOLDERS, DISTRIBUTORS OR SUPPLIERS BE LIABLE TO YOU FOR ANY INDIRECT, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING WITHOUT LIMITATION, SPECIAL, PUNITIVE OR EXEMPLARY DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, OR LOSS OF BUSINESS INFORMATION) ARISING OUT OF OR IN CONNECTION WITH THIS SOFTWARE LICENSE AGREEMENT OR THE SUBJECT MATTER HEREOF EVEN IF CNC HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. CNC'S ENTIRE LIABILITY WITH RESPECT TO ITS OBLIGATIONS UNDER THIS SOFTWARE LICENSE AGREEMENT OR OTHERWISE SHALL NOT EXCEED THE AMOUNT OF THE LICENSE FEE PAID BY YOU FOR THE SOFTWARE. SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF IMPLIED WARRANTIES OR LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THE ABOVE LIMITATIONS OR EXCLUSIONS MAY NOT APPLY TO YOU.

Indemnification

You shall indemnify and hold harmless CNC, its officers, directors, employees, suppliers and agents from and against all losses, settlements, claims, actions, suits, proceedings, judgments, awards, damages, liabilities, costs and expenses including, without limitation, reasonable attorneys' fees (collectively "Losses") which arise out of or as a result of any breach of this Software License Agreement by you or your employees, agents, resellers, dealers or sub-dealers and shall reimburse CNC for any and all legal, accounting and other fees, costs and expenses reasonably incurred

by any of them in connection with investigating, mitigating or defending any such Losses.

Educational Pricing

If this Software was obtained through or in accordance with a CNC "Educational Pricing" plan, option, grant, schedule or program, it may not be used by anyone, including you, to conduct any computer aided design, computer aided drafting, computer aided machining, or training activities that, directly or indirectly, generate or otherwise result in monetary revenues for the benefit of any individual or any entity, other than the school that originally received this Software.

Termination

This Software License Agreement is effective until terminated. You may terminate this Software License Agreement at any time by returning to CNC all copies of the Software under your control and by returning the Security Mechanism to CNC. CNC may terminate this Software License Agreement if CNC determines, in its sole discretion, that you have violated the terms of this Software License Agreement. Upon termination of this Software License Agreement, you agree to immediately return to CNC all copies of the Software, return the Security Mechanism to CNC, and certify to CNC in writing that all known copies, including backup copies, have been returned. All provisions relating to confidentiality, proprietary rights, indemnification and non-disclosure shall survive the termination of this Software License Agreement. **You may not transfer this Software to the purchaser of any equipment on which the Software may be resident. You may not transfer this Software via liquidation, bankruptcy, auction, close of business, or any other method that does not involve an authorized Mastercam reseller. This License is for you alone.** In the event you breach the provisions of this Section, CNC shall be entitled to liquidated damages in the amount of Fifteen Thousand Dollars ($15,000), plus its reasonable attorney's fees and court costs.

General

This Software License Agreement shall be construed, interpreted and governed by the laws of the state of Connecticut, without regard to conflicts of law provisions. The sole jurisdiction and venue for any litigation arising from or related to this Software License Agreement or the subject matter hereof shall be in an appropriate state or federal court located in Hartford, Connecticut. You hereby submit to the personal jurisdiction of the Federal District Court for the District of Connecticut and to the Superior Court of the State of Connecticut. This Software License Agreement shall constitute the entire agreement between you and CNC with respect to the subject matter hereof. Any waiver or modification of this Software License Agreement shall be valid only if it is in writing and signed by both parties hereto. If any part of this Agreement is found invalid or unenforceable by a court of competent jurisdiction, the remainder of this Agreement shall be interpreted so as to reasonably effect the intention of the parties.

U.S. Government Restricted Rights

The Software provided hereunder is a "commercial item," as that term is defined in 48 C.F.R. 2.101, consisting of "commercial computer software" and "commercial computer software documentation," as such terms are used in 48 C.F.R. 12.212. Consistent with 48 C.F.R. 12.212 and 48 C.F.R. 227.7202-1 through 227.7202-4, the Software made available to the United States of America, its agencies and/or instrumentalities, is provided with only those rights set forth in this Agreement. Use, duplication or disclosure of the Software by the government is subject to the restrictions as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software-Restricted Rights clause at 48 C.F.R. 52.227-19, as amended, or any successor regulations thereto.

Export Restrictions

You represent and warrant that you will not, without obtaining prior written authorization from CNC and, if required, of the Bureau of Export Administration of the United States Department of Commerce or other relevant agency of the United States Government, export or re-export, directly or indirectly, the Software from the United States to (i) any country destination to which export is restricted by the Export Administration Regulations of the United States Department of Commerce; (ii) any country subject to sanctions administered by the Office of Foreign Assets Control, United States Department of the Treasury; or (iii) such other countries to which export is

restricted by any other United States government agency. You further agree that you are solely responsible for compliance with any import laws and regulations of the country of destination of a permitted export or re-export, and any other import requirement related to a permitted export or re-export.

Exceptions to Security Mechanism Requirements

CNC software programs Mastercam Design LT and Mastercam Demo do not require the use of Security Mechanisms, and the provisions in this Software License Agreement relating to Security Mechanisms do not apply to your use of such programs, provided, however, that such provisions shall apply to your use of all other Software provided hereunder.

Survival

All provisions of this Software License Agreement relating to confidentiality, non-disclosure, CNC's proprietary rights, disclaimers, and limits of liability, or indemnification by Customer shall survive termination of this License for any reason.

Reservation of Rights

All rights not expressly granted are reserved by CNC.

Trademarks

Mastercam is a registered trademark of CNC.

Windows, Windows 95, Windows 98, Windows NT, and Windows XP are registered trademarks of Microsoft Corporation. Mastercam Verify is created in conjunction with MachineWorks Ltd.

Printed in the United States of America.

Printed on recycled paper.

Contents

1. Tutorials 1

- Introducing Advanced Multiaxis Toolpaths 2
  - Getting help 3
  - Before you begin 4
- Tutorial 1: Parallel Cuts on a Cylinder 5
  - Exercise 1: Creating parallel cutting passes 5
  - Exercise 2: Refining the cutting area 11
  - Exercise 3: Driving the tool with a leading curve 13
  - Tilt angle and tool control 16
    - Exercise 4: Controlling the tool axis 16
  - Gouge checking 20
    - Exercise 5: Gouge prevention by retracting the tool 21
    - Exercise 6: Omitting gouge points from the toolpath 25
  - Approach and retract strategies 29
    - Exercise 7: Creating entry and exit moves 29
    - Exercise 8: Using a fixed tool angle 32
    - Exercise 9: Using a single point for tool axis control 34
    - Exercise 10: Stopping the toolpath on a gouge 36
- Tutorial 2: Multiaxis Toolpath Techniques 38
  - Exercise 1: Creating cuts parallel to a leading curve 39
  - Exercise 2: Creating a swarf toolpath 43
  - Exercise 3: Handling problem areas in the part model 45
  - Exercise 4: Combining several gouge and collision checks 50

2. Application Examples 57

- Example 1: Compressor Armature with 4-Axis Output 58
  - Tooling 59
  - Drive geometry and cutting passes 60
  - Tool axis control and gouge checking 61

- Example 2: Engraving 63
  - Tooling 65
  - Drive geometry and cutting passes 66
  - Linking the cuts 68
  - Tool axis control and gouge checking 70
- Example 3: Camshaft on a Multitasking Lathe 71
  - Tooling 73
  - Drive geometry and cutting passes 74
  - Tool axis control and gouge checking 77
- Example 4: Electrode Machining 79
  - Tooling 80
  - Cutting parameters—semi-finish toolpath 82
    - Drive geometry and cutting passes 82
    - Tool axis control and gouge checking 83
  - Cutting parameters—finish toolpath 85
    - Drive geometry and cutting passes 85
    - Tool axis control and gouge checking 87
- Example 5: Impeller Floor Roughing 91
  - Drive geometry and cutting passes 93
  - Tool axis control 96
  - Roughing options 97
    - Plunge roughing 98
- Example 6: Impeller Blade Semi-finishing 101
  - Tooling 102
  - Drive geometry and cutting passes 103
  - Tool axis control 104
  - Gouge check strategies 106
- Example 7: Impeller Floor Finishing 107
  - Drive geometry and cutting passes 108
  - Tool axis control 110
  - Gouge check strategies 111
- Example 8: Impeller Blade Finishing 113
  - Drive geometry and cutting passes 114
  - Tool axis control 115
  - Gouge check strategies 116

chapter 1

Tutorials

This chapter includes two tutorial examples designed to introduce Mastercam's new advanced multiaxis toolpaths. After completing them, you should know enough about the basic principles to begin applying them to specific applications.

Tutorial 1: Parallel Cuts on a Cylinder

- **Creating parallel cutting passes** . . . . . *page 5*
- **Refining the cutting area** . . . . . *page 11*
- **Driving the tool with a leading curve** . . . . . *page 13*
- **Controlling the tool axis** . . . . . *page 16*
- **Gouge prevention by retracting the tool** . . . *page 21*
- **Omitting gouge points from the toolpath** . . *page 25*
- **Creating entry and exit moves** . . . . . *page 29*
- **Using a fixed tool angle** . . . . . *page 32*
- **Using a single point for tool axis control** . . . *page 34*
- **Stopping the toolpath on a gouge** . . . . . *page 36*

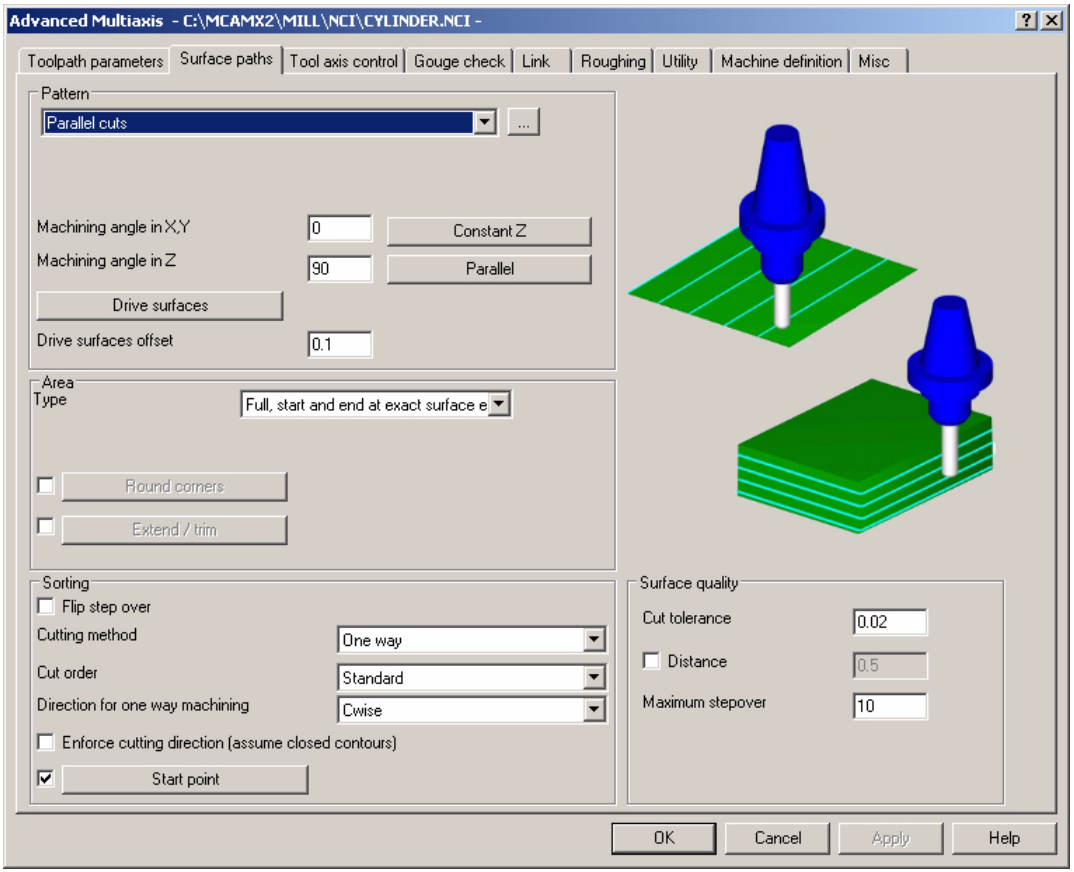
Tutorial 2: Multiaxis Toolpath Techniques

- **Creating cuts parallel to a leading curve** . . . *page 39*
- **Creating a swarf toolpath** . . . . . *page 43*
- **Handling problem areas in the part model** . *page 45*
- **Combining several gouge and collision checks** . . . . . *page 50*

These tutorials assume that you are generally familiar with using Mastercam, including creating and backplotting toolpaths, selecting geometry, working with surfaces, and using the Toolpath Manager. If you have any questions about basic Mastercam functions while working on these exercises, please click the **Help** button on any dialog box or consult the *Mastercam X2 Reference Guide* installed in the \Documentation folder in your Mastercam installation directory.

Introducing Advanced Multiaxis Toolpaths

Mastercam's new advanced multiaxis toolpaths take advantage of a comprehensive, multi-tabbed interface to collect all of your toolpath functions in one place:



This tutorial focuses on the four most important tabs. You will use these tabs for almost every toolpath that you create. Typically you will complete them in order, working from left to right:

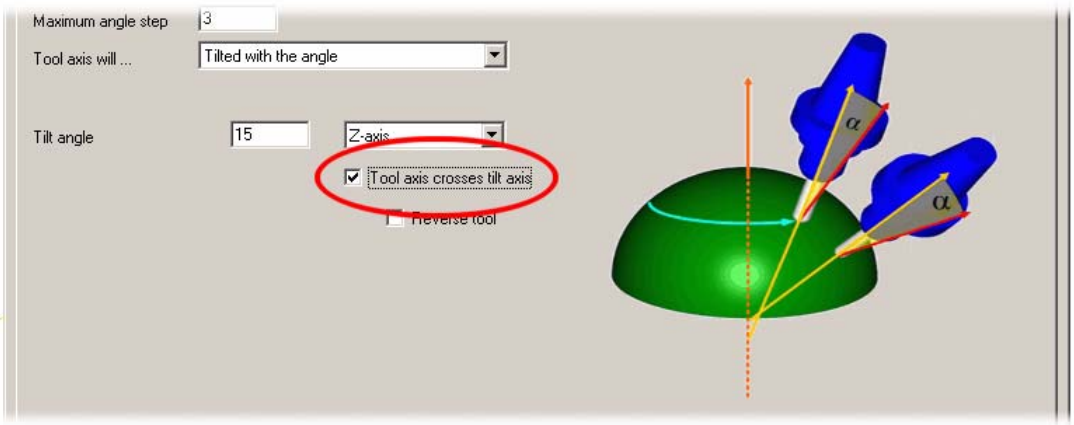
- **Surface paths**—Use this tab to select drive geometry and cut patterns and to organize the cutting passes.

- **Tool axis control**—Use this tab to define the tool orientation and machining limit angles. You can also configure the contact point and choose 3-axis, 4-axis, or 5-axis output.
- **Gouge check**—Use this tab to configure up to four independent gouge- and collision-checking routines. Each individual gouge-check routine can have its own set of check surfaces and geometry and its own type of retract or collision avoidance movement.
- **Link**—Use this tab to configure tool motions for entry and exit moves, moves between passes, and moves that span gaps in your surfaces or toolpaths. You can also specify clearance distances and safety zones.

The advanced multiaxis toolpaths also include a **Toolpath parameters** tab for selecting and managing tools and setting feeds and speeds. This is the same tab used in other Mastercam toolpaths, and so is not described in this tutorial.

Getting help

The advanced multiaxis interface includes many detailed pictures that are linked to individual fields. Simply click in a field to display a visual explanation of it:



Every dialog box includes a main help button in the lower right corner that you can click to learn more.

Before you begin

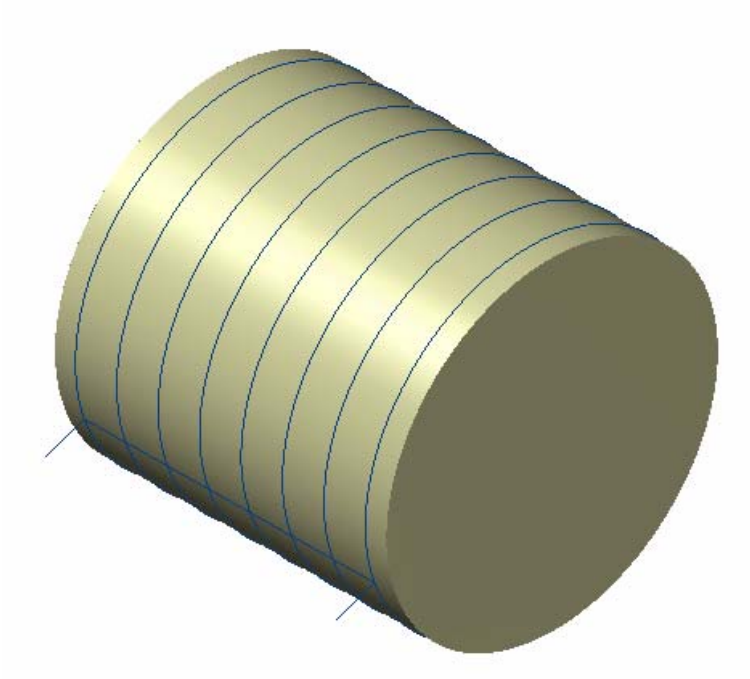
Although these toolpaths are extremely powerful and flexible, there are some important limitations that you should be aware of.

- These toolpaths work on surfaces, solids, and solid faces. Solid selection is available for most advanced multiaxis toolpath strategies, *with the following exceptions*:
  - Toolpaths that require the selection of a defined edge (solid edge)
  - Toolpaths that require the selection of only a single surface (solid face)
- The orientation of the toolpath with respect to your surfaces is determined by the surface normal. Before creating the toolpath, make sure that your normals point in the proper direction. If necessary, use the **Edit, Change normals** function before creating the toolpath. (You can also change the normals after creating the toolpath and simply regenerate the toolpath.)
- The defaults for these toolpaths are hard-coded and are not stored in the .defaults file with your other toolpath defaults.

Tutorial 1: Parallel Cuts on a Cylinder

1

The first tutorial is a simple toolpath that creates parallel cuts around a cylinder, as shown in the following picture. It measures 80mm long, with a radius of 40mm, with its long axis parallel to the X axis. You will use a 30mm ball endmill.



The part geometry and tool are saved in the following file:

\Documentation\ExampleParts\Multiaxis_Example1.mcx

Load this part to begin the tutorial.

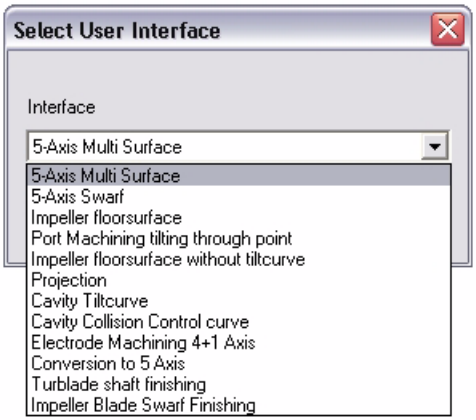
Note: All the parts and tools in this tutorial are metric.

Exercise 1: Creating parallel cutting passes

▶ **Starting the advanced multiaxis toolpaths**

1 From the **Toolpaths** menu, choose **Advanced Multiaxis**.

The **Select User Interface** dialog box displays. It gives you the option of selecting one of several application-specific interfaces. These are simplified interfaces that let you quickly create toolpaths for specific needs; most of the options are set automatically "behind the scenes" and are hidden from you. For this tutorial, you will use **5-Axis Multi Surface**. This is the main interface that exposes all of the multiaxis options, so it provides a better introduction to the advanced multiaxis capabilities.

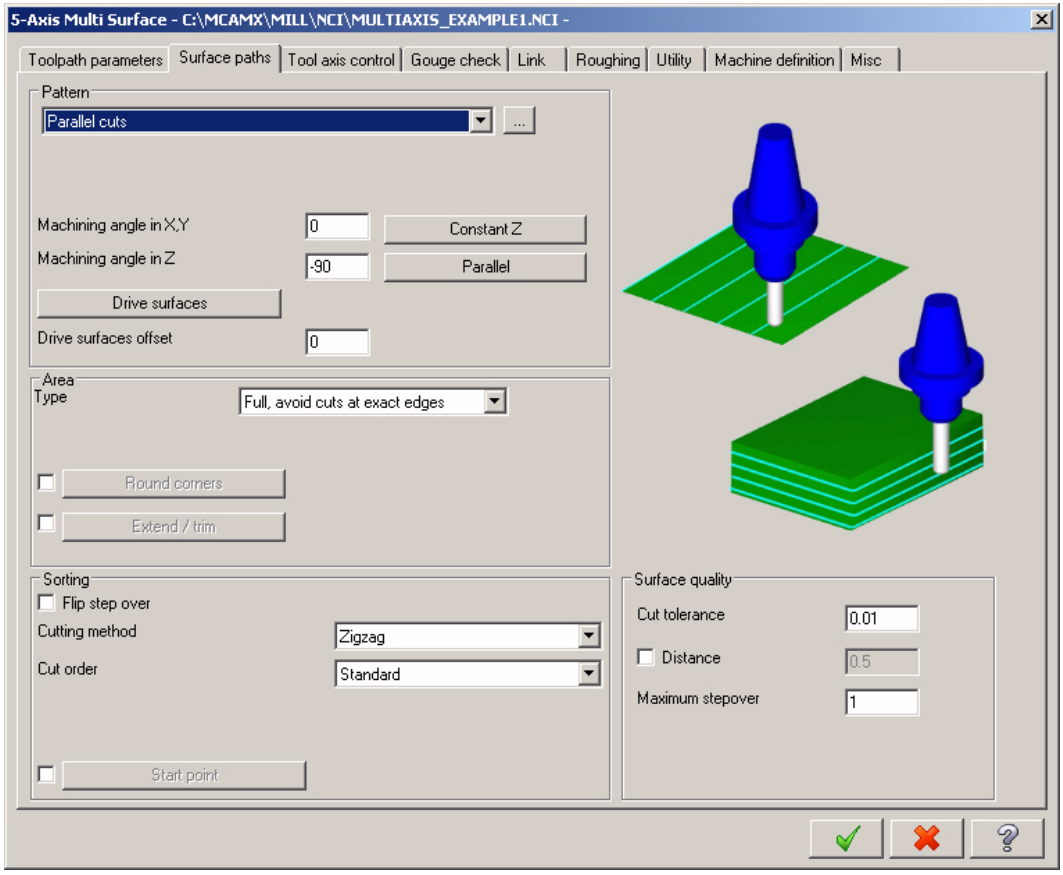


2 Select **5-Axis Multi Surface** and click **OK**.

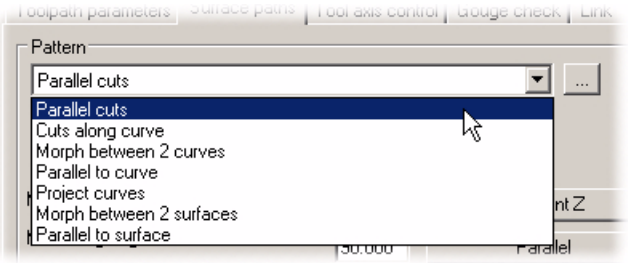
3 On the **Toolpath Parameters** tab, choose the 30mm ball endmill.

▶ Selecting a cutting pattern

1 Select the **Surface paths** tab.

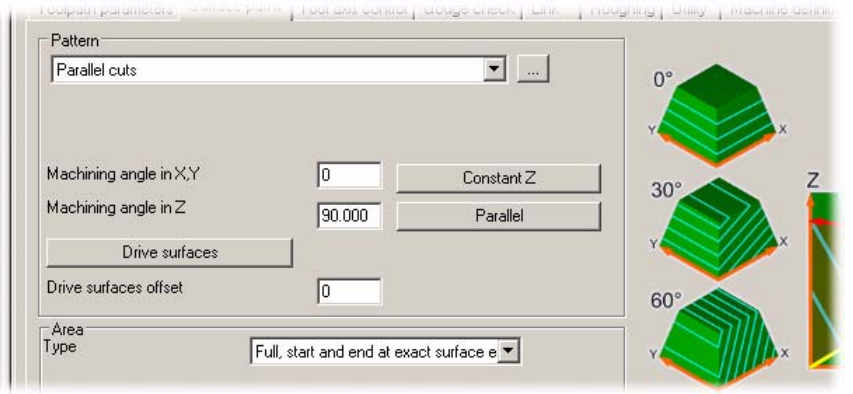


2 Select **Parallel cuts** as the cutting pattern.



3 Set the **Machining angle in X,Y** to **0**, and the **Machining angle in Z** to **90**.

The other options that are available in the **Pattern** section depend on the selected cut pattern. For parallel cuts, the most important parameters are the angle and orientation of the cuts.



- Use **Machining angle in Z** to orient the tool plane in which the cuts are created. A value of **0** creates cuts parallel to the XY plane, while **90** creates cuts perpendicular to it.
- Use **Machining angle in X,Y** to rotate the orientation of the cuts within the cutting plane. A value of **0** creates cuts aligned with the Y axis, while **90** creates cuts aligned with the X axis.

You can type values directly in the fields or use the other buttons to pre-set values for typical applications:

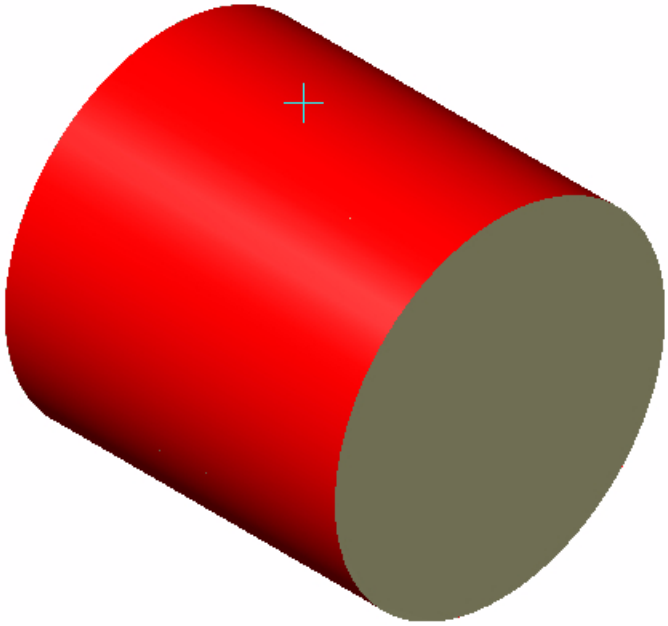
- Click **Constant Z** to create waterline-style passes along the surfaces.
- Click **Parallel** to orient the cutting passes perpendicular to the XY plane.
- Click the **[…]** button to select other planar orientations. Mastercam automatically fills in the machining angles to match the selected plane.

For this application, all you need to do is click **Parallel** to set the proper machining angles.

▶ Selecting the drive surfaces

1 Click the **Drive surfaces** button.

2 Select the red surface shown below.



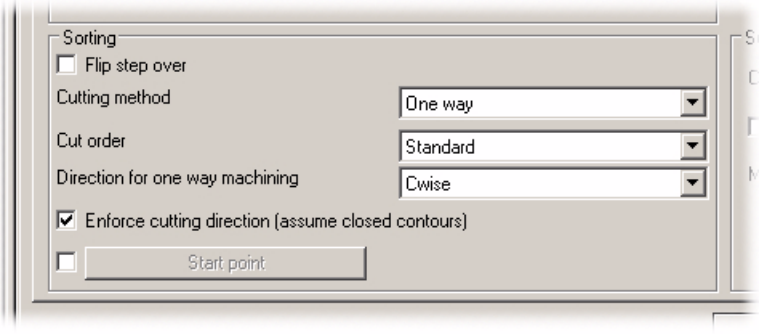
3 Enter **0** for **Drive surfaces offset**.

- To leave stock on the drive surface, enter a positive value here.
- To cut below the drive surface, enter a negative value.

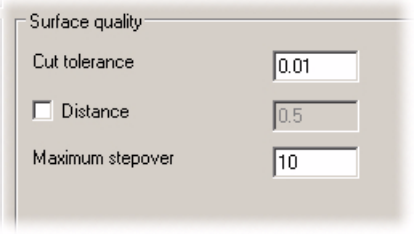
▶ Organizing the cutting passes

1 From the **Cutting method** list, select **One way**.

This makes the tool run around the cylinder in one direction. Using **Zigzag** would alternate the cutting direction with every cut.

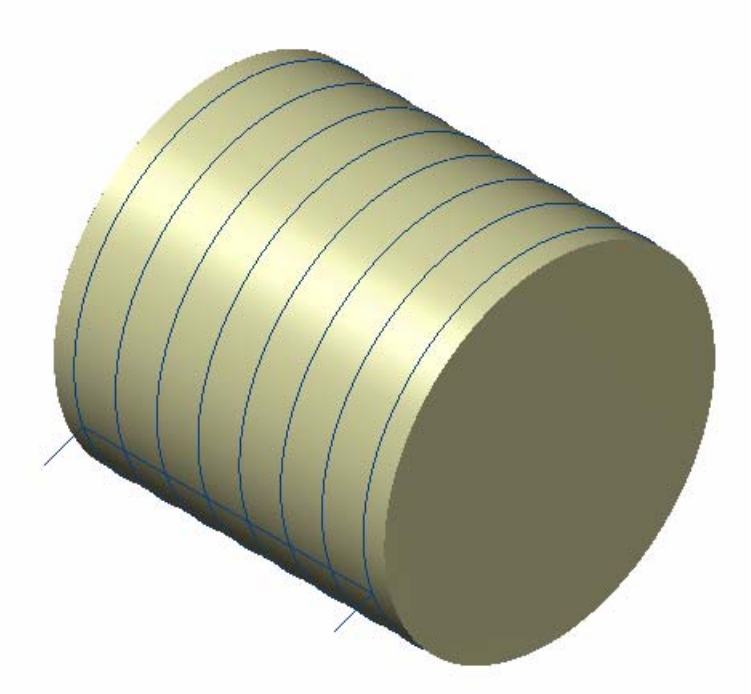


2 Set the **Maximum stepover** to **10**. That sets the distance between every slice to 10mm.



3 Click **OK** to generate the tool path.

Your cutting passes should look like the following picture:



Note: For clarity's sake, most of the toolpath pictures in this tutorial are taken while backplotting.

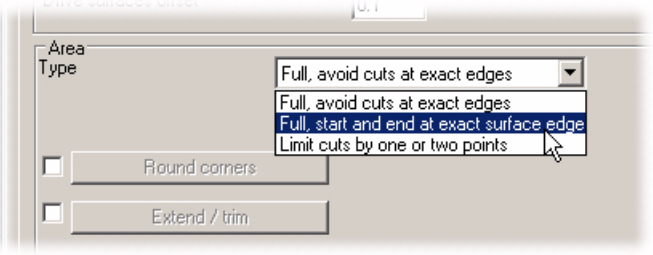
Exercise 2: Refining the cutting area

In the toolpath above, notice that there are gaps between the edge of the model and the first and last cutting passes. The gap is approximately half the maximum stepover amount. In this exercise, you will modify the toolpath to eliminate them. You will also start the toolpath at the top of the cylinder instead of the side.

▶ Expanding the cutting area

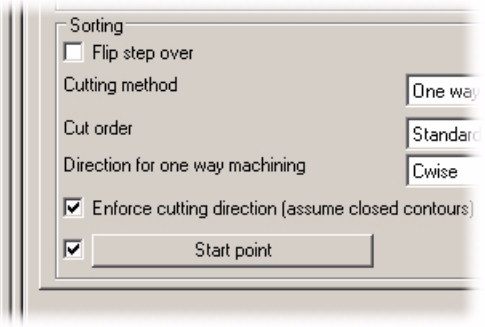
1. If necessary, click the **Parameters** icon in the Toolpath Manager to display the settings for the existing operation.

2 On the **Surface paths** page, change the **Area Type** to **Full start and end at exact surface edge**.

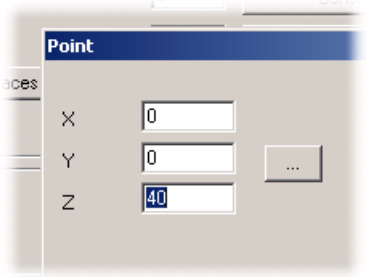

Area
Type
Full, avoid cuts at exact edges
Full, avoid cuts at exact edges
Full, start and end at exact surface edge
Limit cuts by one or two points
Round corners
Extend / trim

Use these options to either explicitly include the edge of the surfaces, or exclude them. For example, if there are problems on the edges of your surfaces, calculating a cutting pass slightly away from the exact edge might reduce chatter and produce a cleaner toolpath.

3 Click the **Start point** check box and button.

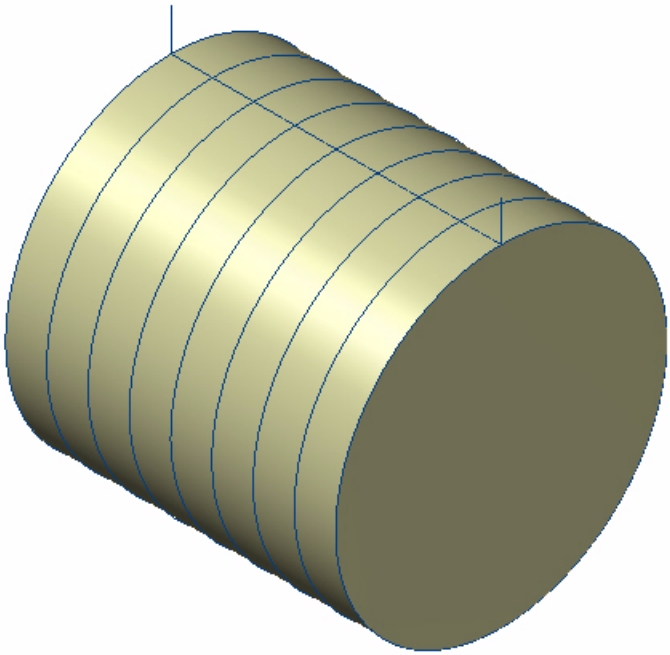


4 Enter a start point of **X0**, **Y0**, **Z40**.



Note: Mastercam uses this point as a hint or guideline only. It tries to start the toolpath close to this point, subject to the other drive surface and cut pattern settings.

5 Click **OK** and then regenerate the toolpath. It should look like the following picture.



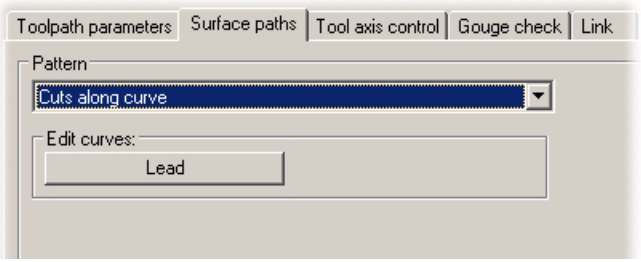
Exercise 3: Driving the tool with a leading curve

In this exercise, the cutting pattern will be based on a leading curve. Mastercam creates cuts normal to the leading curve and projects them onto the drive surfaces. In this case, the curve will be a line through the middle of the cylinder. For this part, this produces results similar to the previous exercise, but without needing to set machining angles. This technique is often used when the cylinder is a more complex shape, such as a head port.

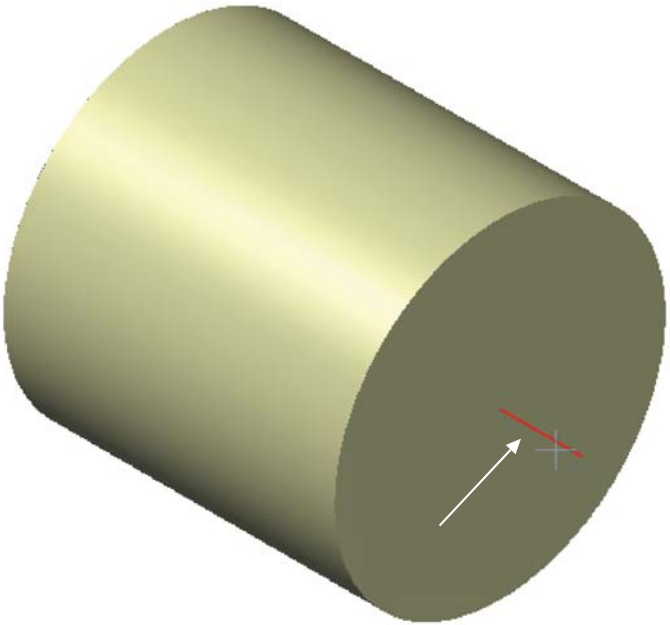
Note: The line that you will use as a drive curve is on level 3 in your part file. If necessary, make this level visible to display the line.

▶ Selecting the leading curve

1. Make a copy of the toolpath from the previous exercises. (You will edit it again in later exercises.)
2. Click the **Parameters** icon in the Toolpath Manager for the new operation.
3. On the **Surface paths** page, change the **Pattern** to **Cuts along curve**.



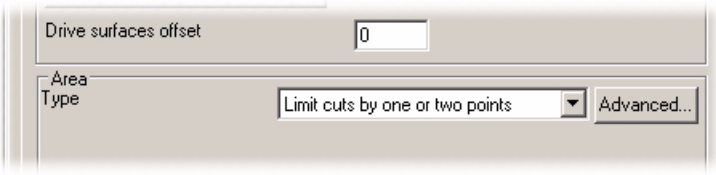
4. Click the **Lead** button.
5. Select the line as shown in the following picture. The chaining direction of the selected curve determines where the toolpath will begin.



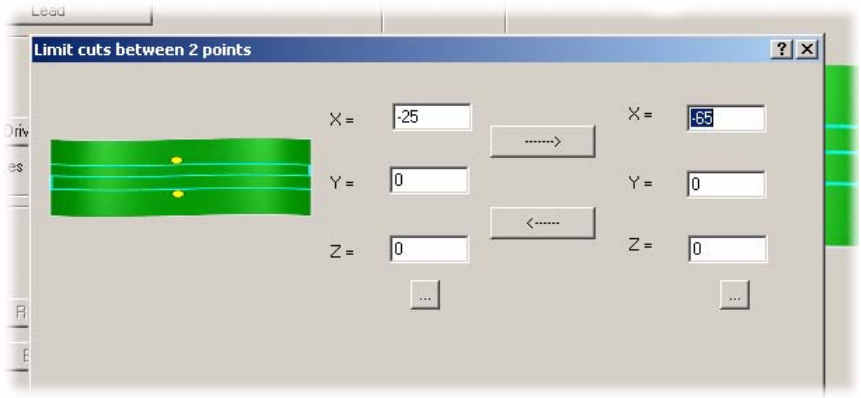
▶ Limiting the cutting area between two points

For this toolpath, you will limit the cutting area to the area between two points. This lets you limit the cutting zone to a specific portion of the cylinder.

1 From the Area: Type list, select **Limit cuts by one or two points**.



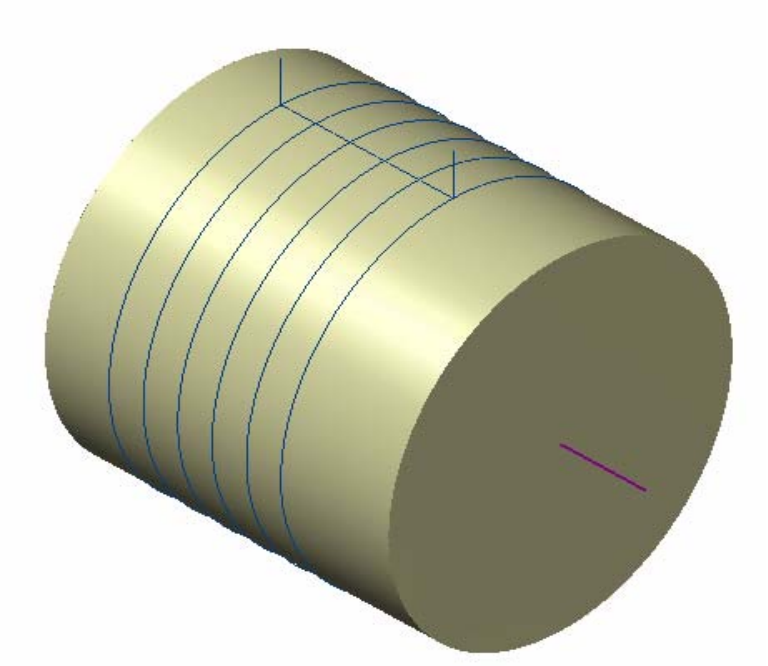
2 Click the **Advanced** button and enter the limits **X–25** and **X–65**. Mastercam will create the toolpath between these two points only.



TIP: If the two points are the same, Mastercam will create a single pass through the point.

3 Click **OK** to close all the dialog boxes and then regenerate the tool path.

Your cutting passes should look like the following picture:



Tilt angle and tool control

The **Tool axis control** tab lets you define the tool axis relative to the drive geometry, cutting direction, guide or construction geometry, or a combination of them. You can also specify:

- lead/lag angles, side tilt angles, and angle limits
- tool tip and contact point, including user-defined locations on the tool tip
- output type (3-axis, 4-axis, or 5-axis)

Exercise 4: Controlling the tool axis

For this exercise, you will modify the toolpath that you created in Exercises 1 and 2.

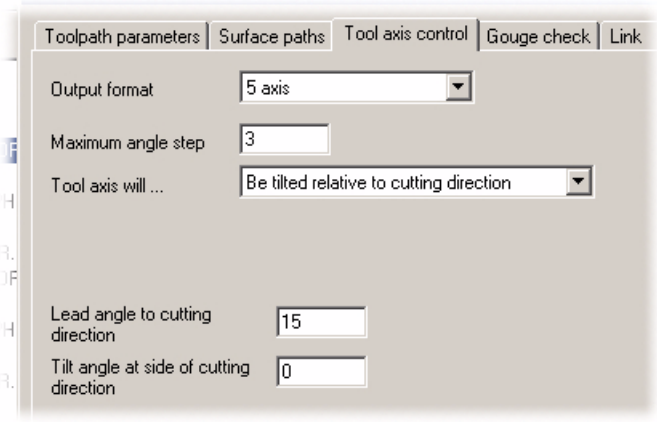
▶ Applying lead/lag angles

1. Click the **Parameters** icon in the Toolpath Manager for Operation 1.

2 Select the **Tool axis control** tab.

3 Select the **Be tilted relative to cutting direction** strategy.

4 Enter a **Lead angle** of **15** degrees.

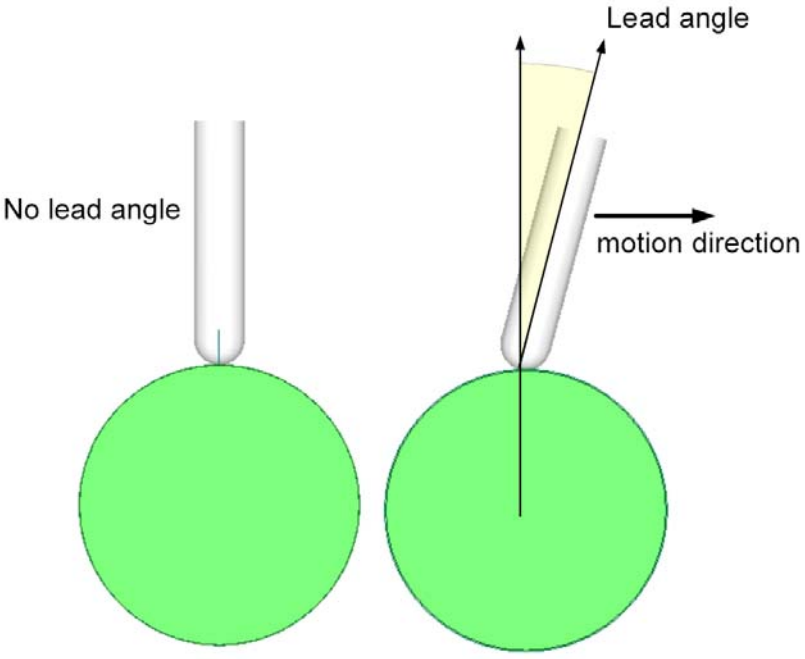


5 Click **OK** to close all the dialog boxes and then regenerate the tool path.

The original setting of the tool axis was fixed and normal to the surfaces. With the new tool axis strategy, you can define a lead/lag angle to cutting direction.

- Positive value = lead angle (tool leans to the movement direction)
- Negative values = lag angle (tool leans away from the movement direction).

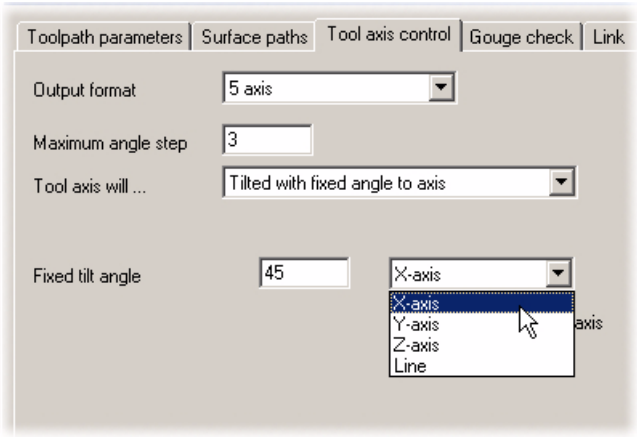
When you backplot the toolpath from the right side, the lead angle should appear as follows:



▶ **Applying side tilt angles**

1. Click the **Parameters** icon and select the **Tool axis control** tab.
2. Select the **Tilted with fixed angle to axis** strategy.

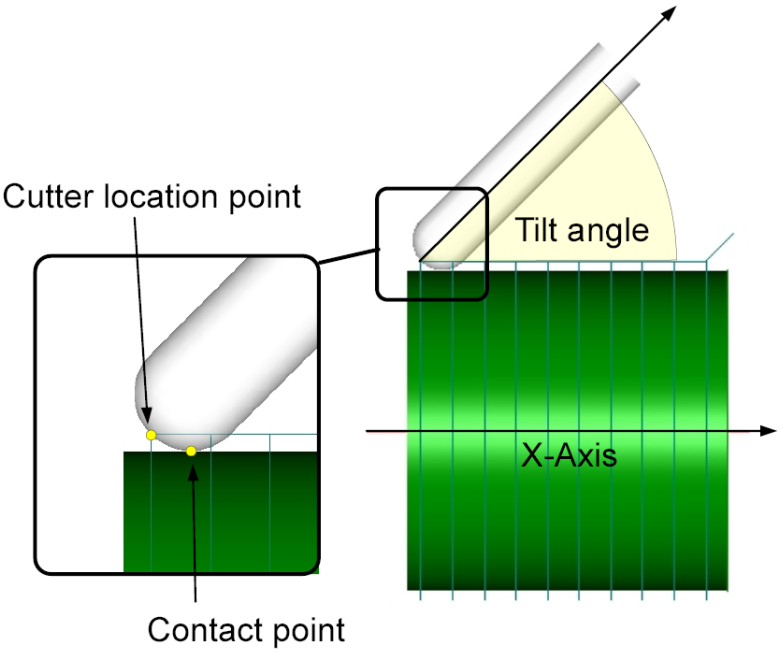
3 Enter a **Fixed tilt angle** of **45** degrees, relative to the **X axis,** as shown in the following picture.



4 Click **OK** to close all the dialog boxes and then regenerate the tool path.

Use this option to set the tool axis at a fixed angle to any axis or to a line that you choose. The tilt axis and surface normal build a plane in which the tool tilts. Notice that Mastercam still calculates the location of the cutting pass based on the

tool tip, but offset from the surface to maintain the proper contact point



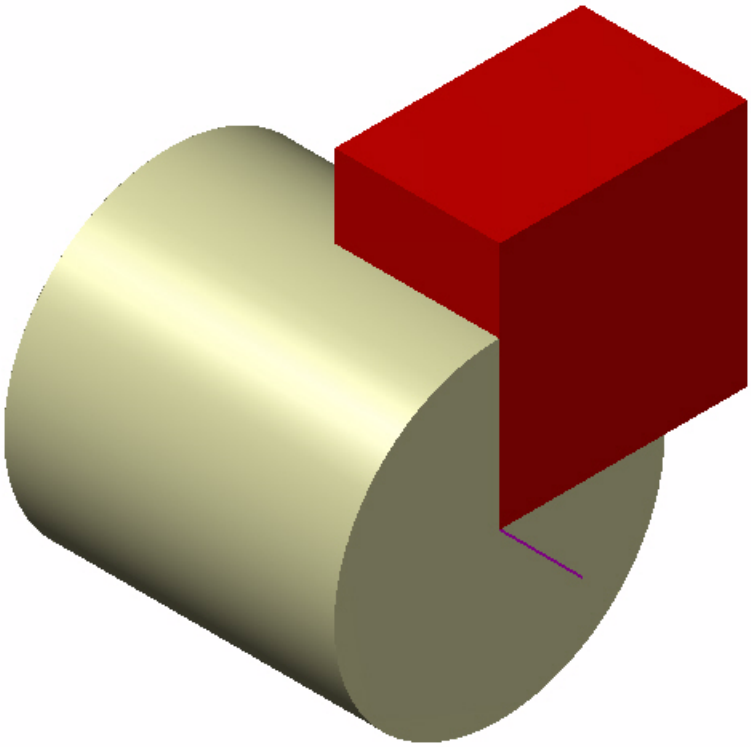
The tool will stay tilted towards X+ in a constant angle of 45 degrees all along the tool path.

Gouge checking

The **Gouge check** tab lets you define up to four independent gouge and collision checks to be applied to your toolpath. Each one has three main components:

- the parts of the tool to check—tool tip, shaft, arbor, holder, or any combination.
- the type of action to take when a collision is detected—retract away from the toolpath, tilt the tool away, omit the points completely, or stop processing the toolpath. For each option, additional parameters let you configure the tool motion.
- the geometry to check against. You can select drive surfaces, check surfaces, or both. For maximum flexibility, each individual test has its own set of geometry, and this can be completely independent of the surfaces selected in the **Surface paths** tab.

To prepare for this exercise, make level 2 visible. This contains the check surfaces that you will collision-check against. Your part should look like the following picture:

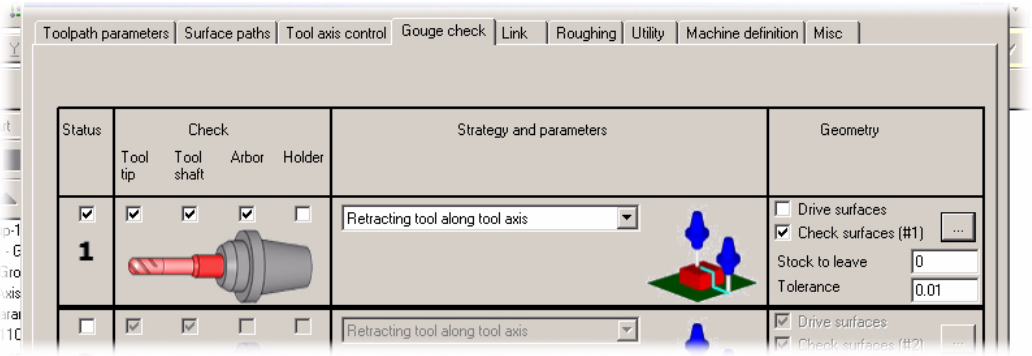


Exercise 5: Gouge prevention by retracting the tool

▶ **Selecting and applying a gouge-check strategy**

1. Click the **Parameters** icon for Operation 1.
2. Turn off the tool axis options from the last exercise. Select the **Tool axis control** tab and select the **Not be tilted and stays normal to surface** strategy.
3. Select the **Gouge check** tab.

4 Activate gouge check number 1 by selecting it in the **Status** column.



5 Select the **Tool tip**, **Tool shaft**, and **Arbor** to check against.

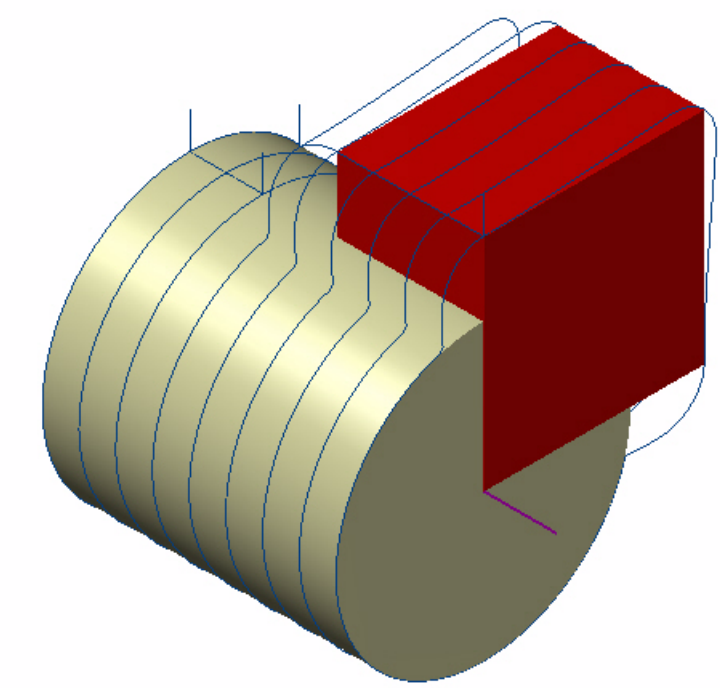
TIP: You can define individual clearance values for each tool component by clicking the **Clearances for tool parts** button.

6 Select the **Retracting tool along tool axis** strategy. This will retract the tool along its axis until the gouge condition is no longer present.

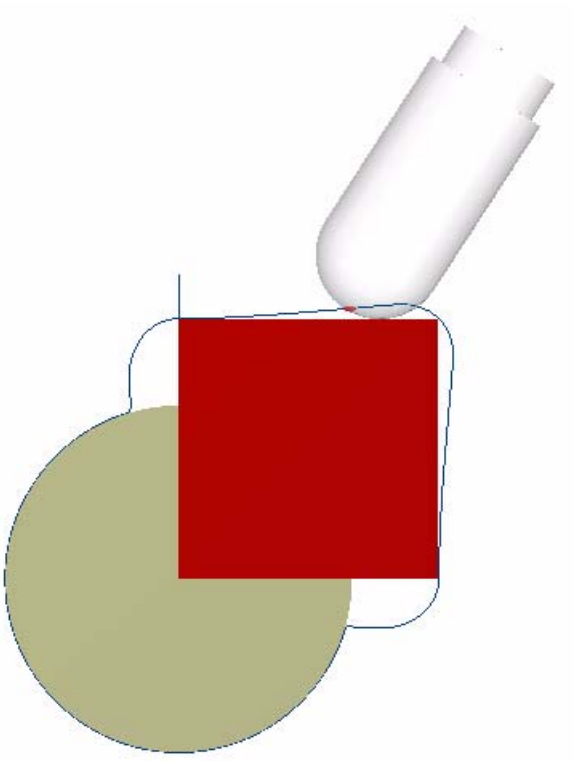
7 Since the purpose of this test is to check against the red box, de-select the **Drive surfaces** option and select **Check surfaces**. Click the **[…]** button and select all the red surfaces.

8 If necessary, deselect the **Check gouge between positions** option.

9 Click **OK** and then regenerate the toolpath. It should look like the following picture.



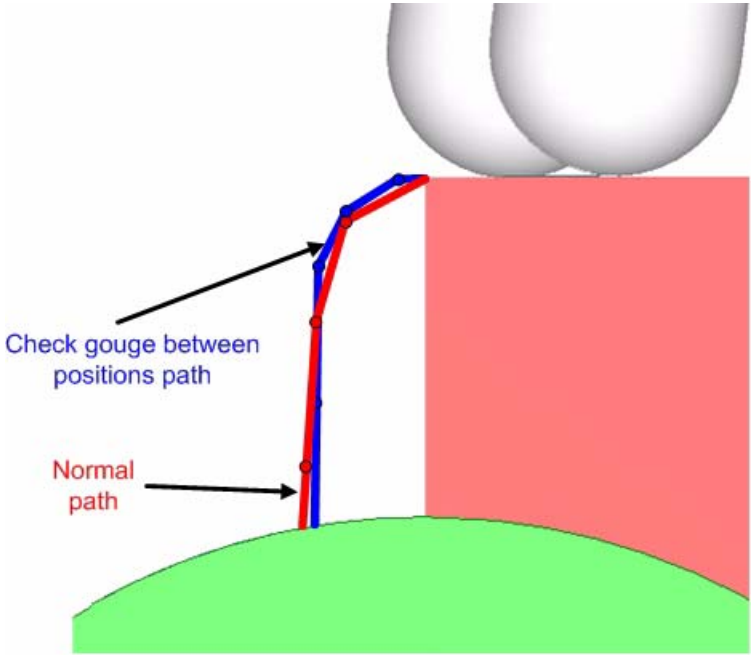
Even while the tool is passing around the check surfaces, the tool axis is still normal to the drive surface, as set in the **Tool axis control** tab.



▶ **Gouge checking between positions**

1. Make a copy of the operation.
2. Go to the **Gouge check** tab for the new copy and select the **Check gouge between positions** option.
3. Click **OK** and then regenerate the toolpath.

When you backplot the toolpath and compare it to the previous operation, you should see a subtle difference as the tool rounds the sharp corner of the check surface.



Selecting this option causes Mastercam to insert additional nodes in the toolpath that result in more accurate gouge checking around sharp corners and other sharp or discontinuous areas of your model. Selecting this option results in increased toolpath processing time, but can produce more accurate gouge checking.

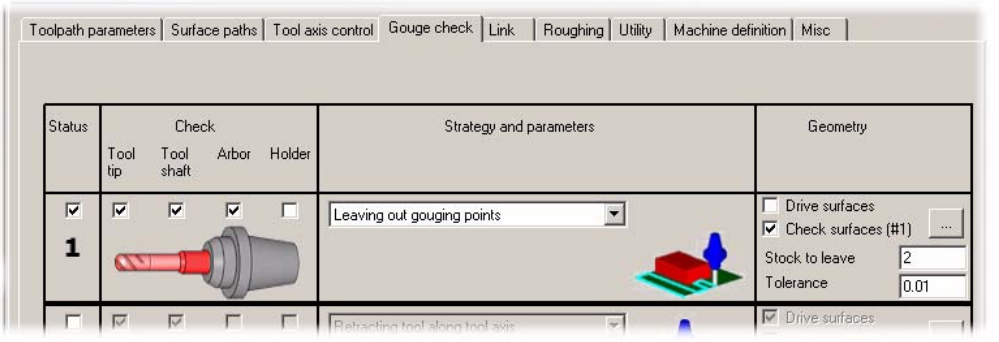
Exercise 6: Omitting gouge points from the toolpath

In the previous exercise, the tool is effectively machining the check surface, since there is 0 stock to leave on those surfaces. In this exercise, you will apply a different retract strategy to the check surfaces. The goal will be to prevent the machining over the check surfaces and maintain a minimum 2mm distance from them.

▶ Applying the "Leaving out gouging points" strategy

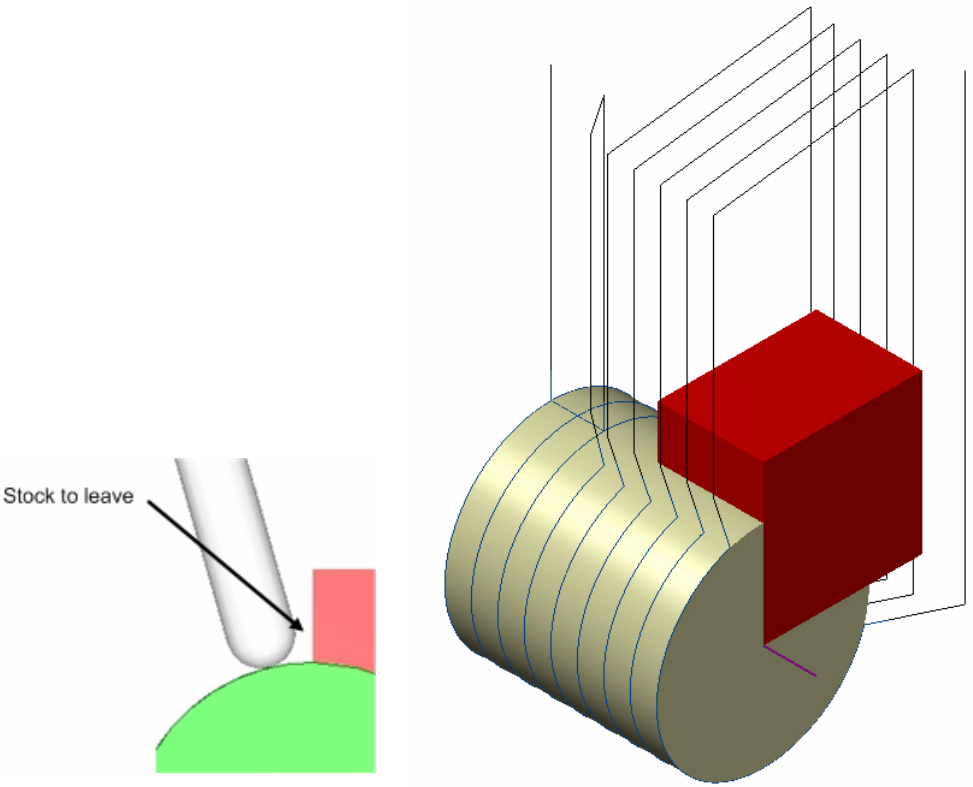
1. Open the toolpath parameters for the current operation.
2. Go to the **Gouge check** tab and change the gouge strategy to **Leaving out gouging points**.

3 Set the **Stock to leave** for the check surfaces as **2**. Your options should look like the following picture.



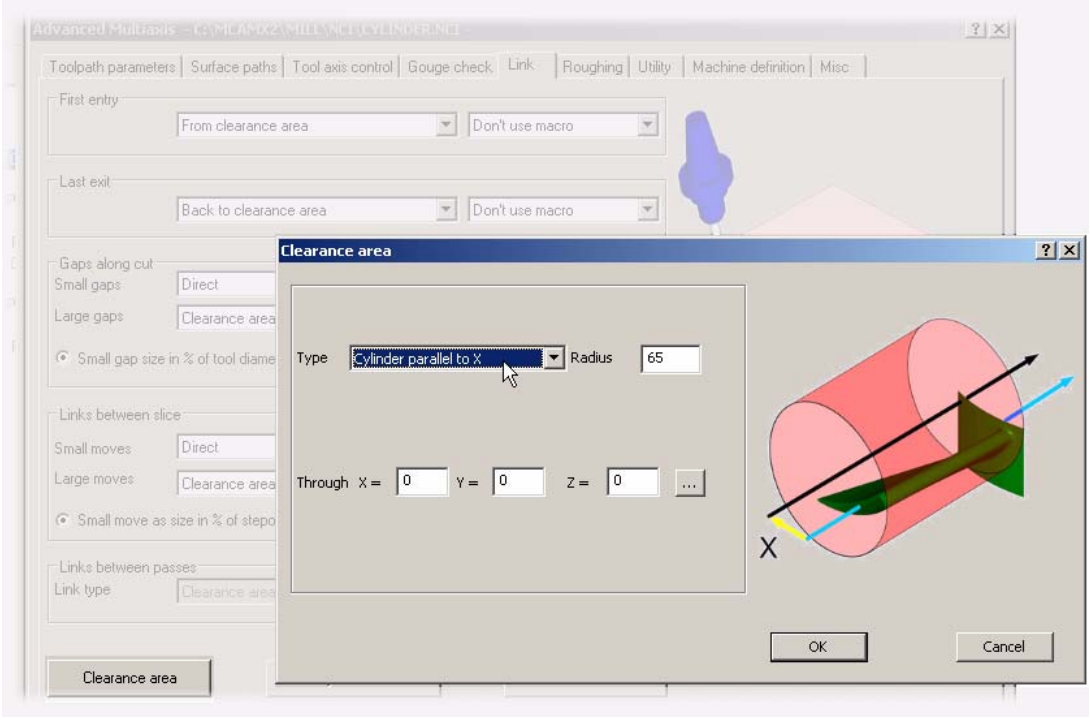
4 Click **OK** and then regenerate the toolpath.

You can see that now when the tool reaches a check surface, it retracts to a clearance plane. When it approaches the part, it still maintains a minimum 2mm from the check surface. In the next procedure you will define a better clearance zone.



▶ Defining clearance planes and safety zones

1. Open the toolpath parameters for the operation and select the **Link** tab. Use the **Link** tab to configure non-cutting tool movements between cutting passes, as well as entry and retract moves.
2. Click the **Clearance area** button.
3. From the **Type** list, select **Cylinder parallel to X** and enter a radius of **65**. Enter anchor point coordinates of **X=0**, **Y=0**, and **Z=0**. This establishes a clearance plane which wraps around the part 65mm from the centerline.



4. Click **OK** to close this dialog box, but leave the **Link** tab on the screen.

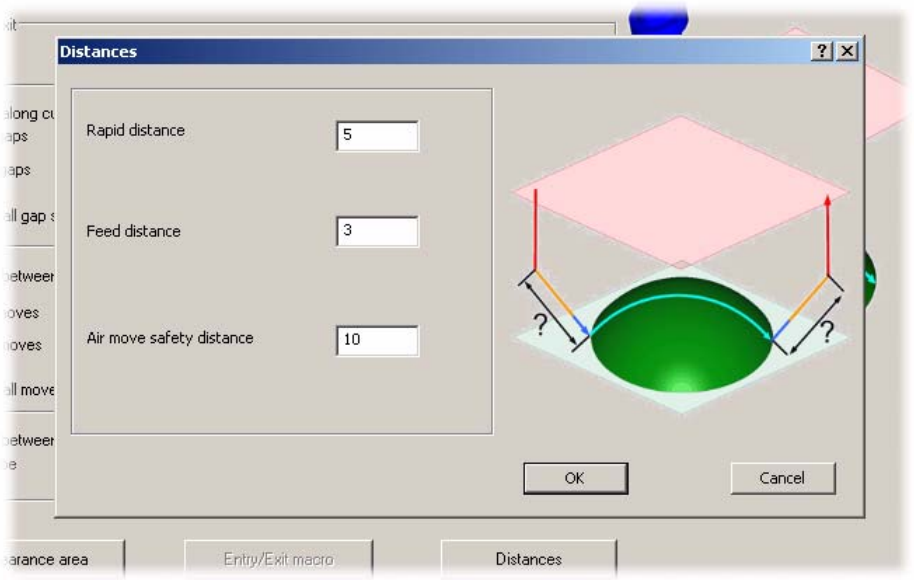
▶ Defining the motion to and from the clearance plane

1. Click the **Distances** button.
2. Enter **5** for the **Rapid distance**. This establishes an intermediate point to which the tool will rapid from the clearance plane.

3 Enter **3** for the **Feed distance**. Once the tool reaches this distance from the surface, it will feed towards the part.

Both distances are measured along the tool axis from the surface being machined.

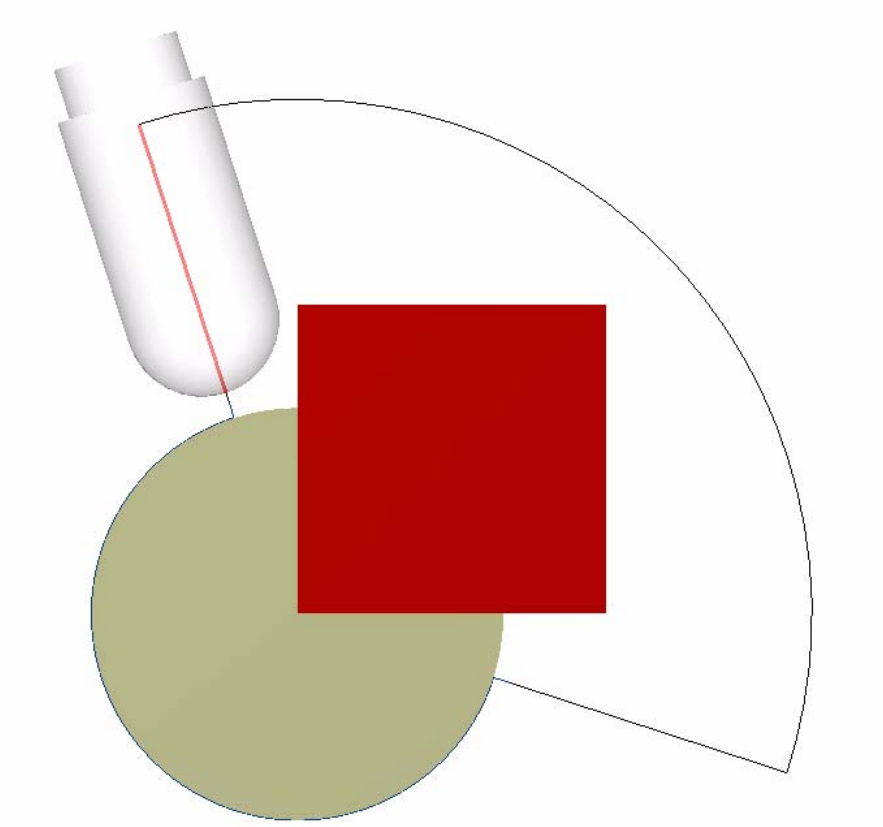
4 Enter **10** as the **Air move safety distance**. As the tool moves across the clearance area, Mastercam ensures that the tool will not come closer to the check surfaces than this distance.



5 Click **OK** and then regenerate the toolpath. You can see that the tool movements around the check surfaces are much more efficient and appropriate for this part.

Notice that the radius of the clearance cylinder is larger than the programmed radius of 65mm. Mastercam automatically increases the

size so that the tool moves safely around the check surfaces, while maintaining the air move safety clearance.



Approach and retract strategies

This section introduces you to different combinations of tool axis control, gouge check, and link settings that you can use to bring the tool on and off the part and retract from check surfaces.

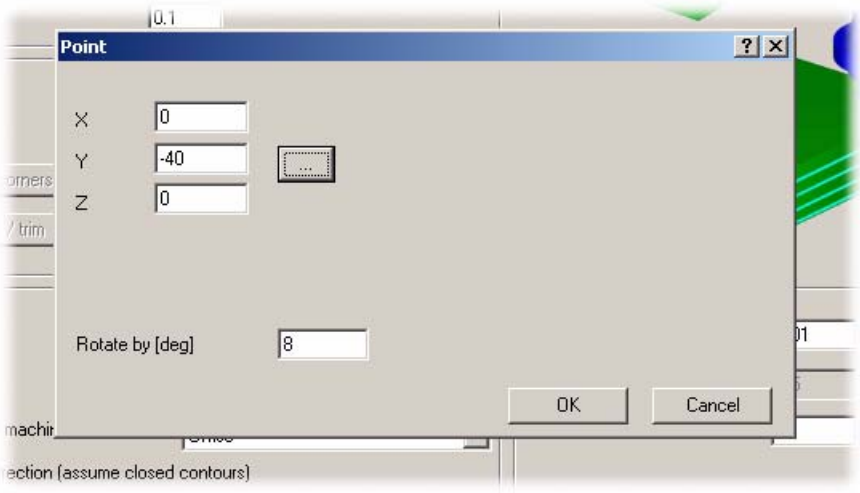
Exercise 7: Creating entry and exit moves

▶ **Adjusting the start point**

1. Open the toolpath parameters for the operation and select the **Surface paths** tab.
2. Click the **Start point** button.

3 Change the start point coordinates to **X0**, **Y–40**, and **Z0**, so that the tool will start machining at the side of the part instead of the top.

4 Enter a **Rotate by (Deg)** angle of **8** and click **OK.** This rotates the starting point of each successive cutting pass by 8 degrees.

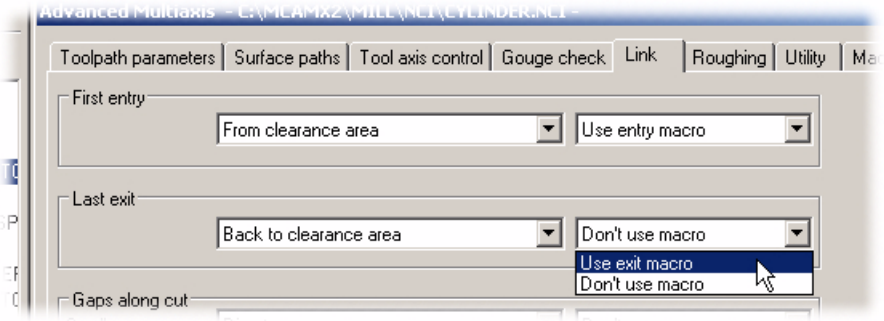


▶ **Creating an entry move**

1 Go to the **Link** tab.

2 In the **First entry** area, set the entry options to **From clearance area** and **Use entry macro**.

3 In the **Last exit** area, set the entry options to **Back to clearance area** and **Use exit macro**.



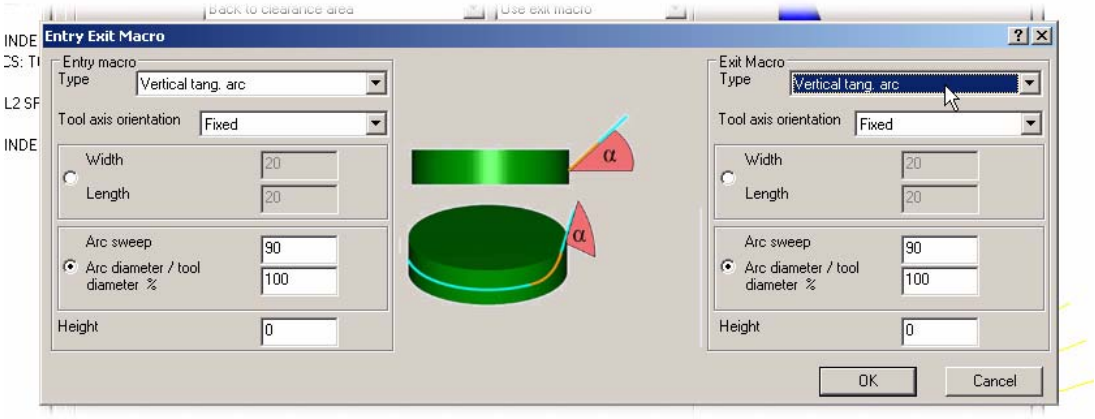
In this case, the word “macro” simply means that you will configure a series of tool motions to approach and retract

from the part, instead of the simple straight linear moves that you have been using so far.

4 Click the **Entry/Exit macro** button to begin defining the moves.

5 Select **Vertical tang. arc** for both the entry and exit macro. This will create an arc move aligned with the parallel cutting passes.

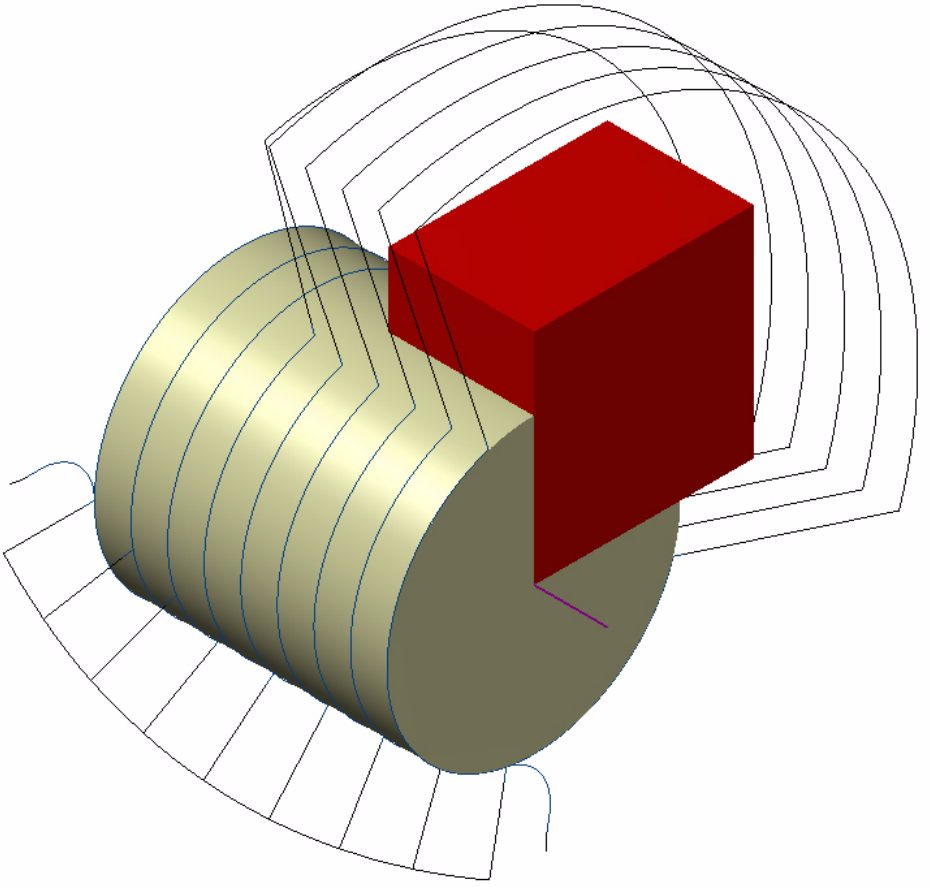
6 Set the **Arc diameter** to **100**% for both arcs.



7 Click **OK** and then regenerate the toolpath.

Your toolpath should look like the following picture. The tangential arcs are applied to the first and last cutting passes only. In this case, the **Rapid distance** and **Feed distance** set in the previous exercise are added to the beginning and end of the arc. You can see that between

each pass, the tool retracts to the clearance plane while Mastercam shifts the start of the next cutting pass by 8 degrees.

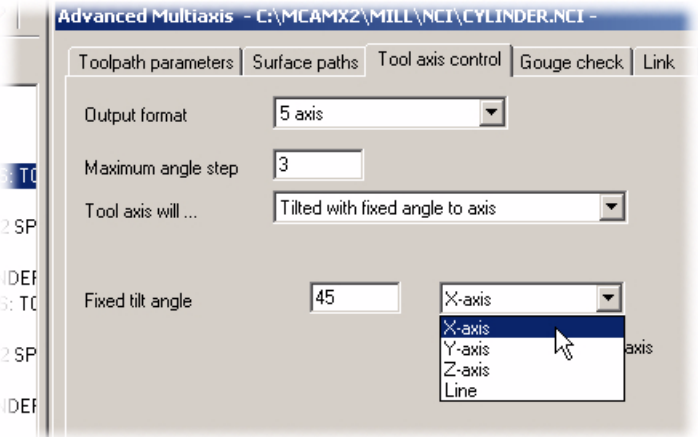


Exercise 8: Using a fixed tool angle

▶ **Locking the tool axis to a fixed angle**

1. Open the toolpath parameters for the operation and select the **Tool axis control** tab.
2. Select the **Tilted with fixed angle to axis** strategy.

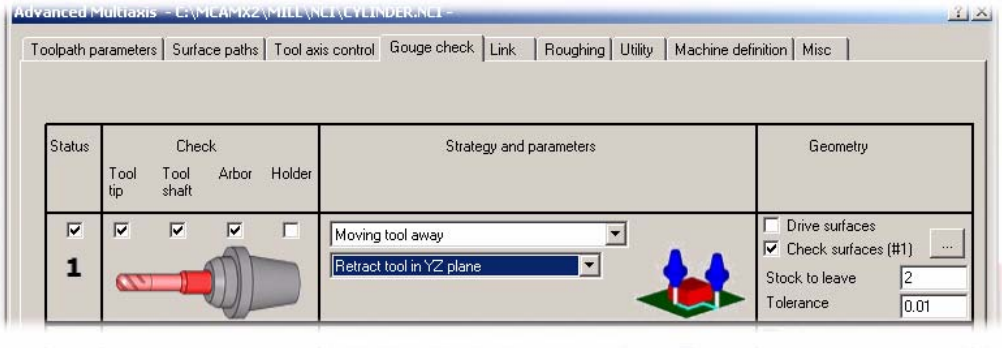
3 Select **X-axis** and set the **Fixed tilt angle** to **45**.



▶ **Locking the retract moves to fixed plane**

1 Go to the **Gouge check** tab.

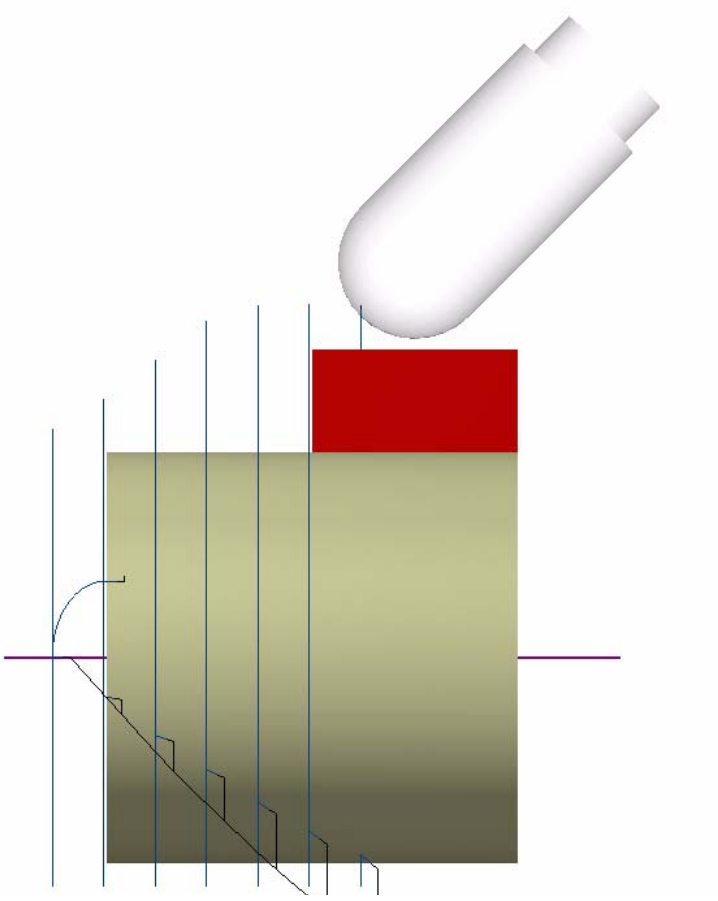
2 Change the strategy to **Moving tool away** and **Retract tool in YZ plane**. This will lock the retract moves to a constant X value.



3 Click **OK** and then regenerate the toolpath.

You can see that even though the retract moves are locked to a constant Z, Mastercam maintains the 45-degree tilt in the tool axis. With this retract strategy, the tool does not pull all the way out to the

clearance plane, but just enough to maintain the **Stock to leave** amount.

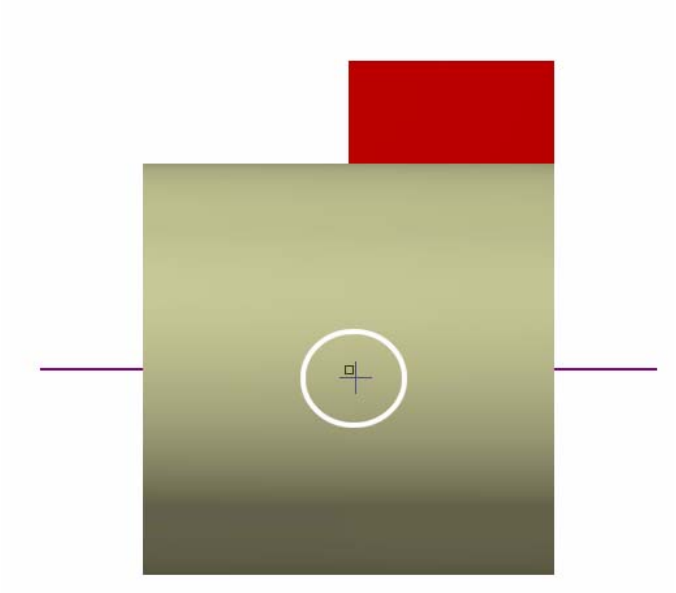


Exercise 9: Using a single point for tool axis control

▶ Tilting the tool axis to a point

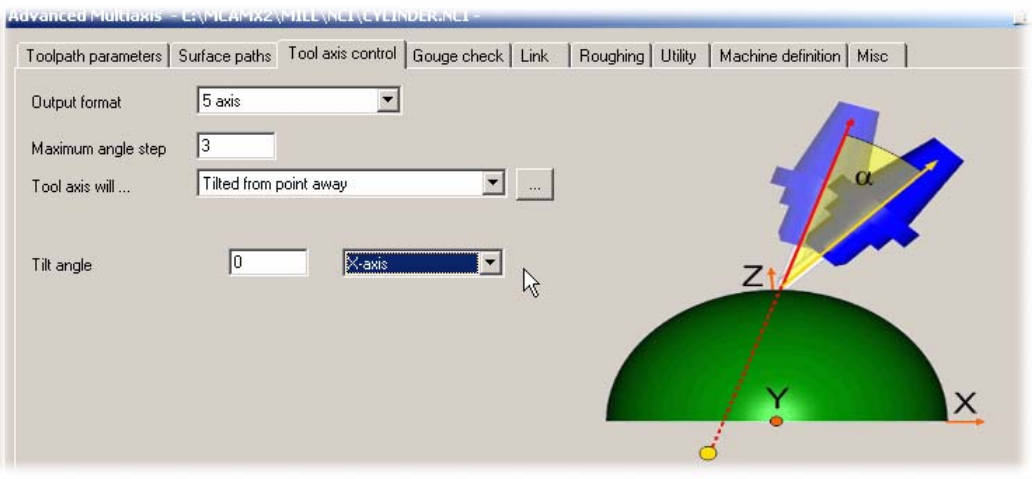
1. Open the toolpath parameters for the operation and select the **Tool axis control** tab.
2. Select the **Tilted from point away** strategy.
3. Click the […] button to select the point, and click it again to return to the graphics window.

4 Select the midpoint of the purple line that runs along the cylinder axis.

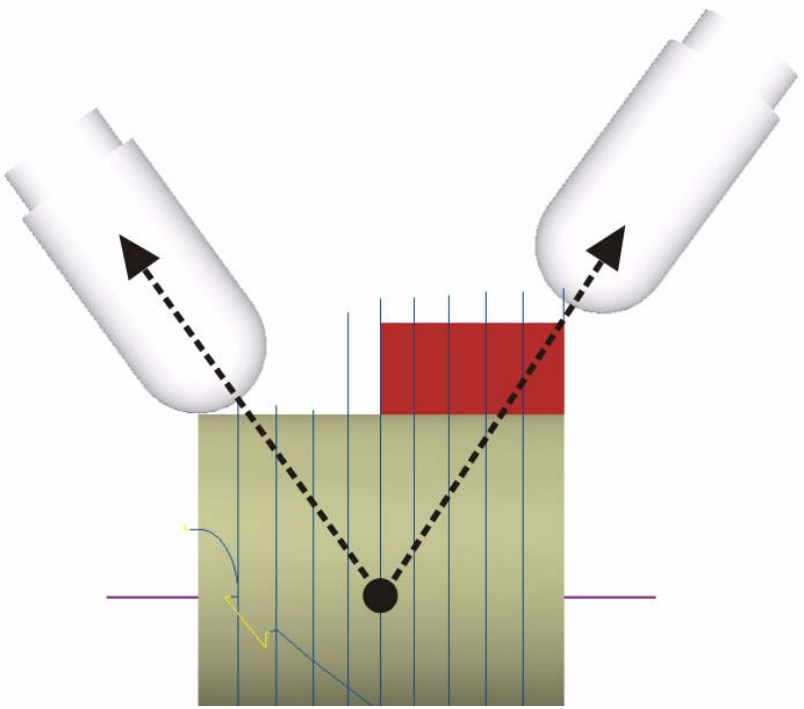


5 Mastercam fills in **X–40** in the Point dialog box. Click **OK** to close it.

6 Set the **Tilt angle** to **0** and select the **X-axis**. Even though you have chosen to base the tool axis on the tilt angle from the point, Mastercam lets you enter an additional angle to add to it.



7 Click **OK** and then regenerate the toolpath. Notice that at all times the tool axis is pointing away from the point.



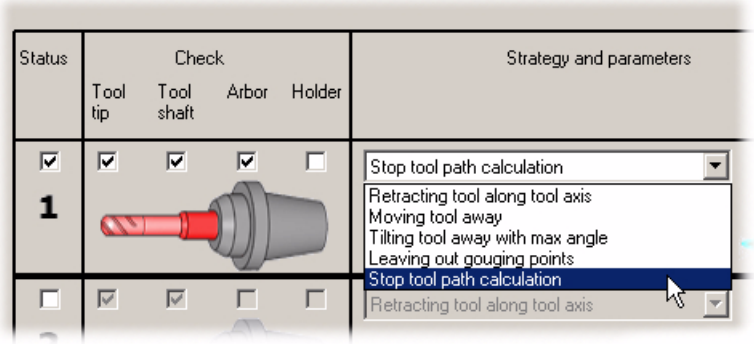
TIP: Mastercam also includes a **Tilt through point** option that you can use when the point is outside the part.

Exercise 10: Stopping the toolpath on a gouge

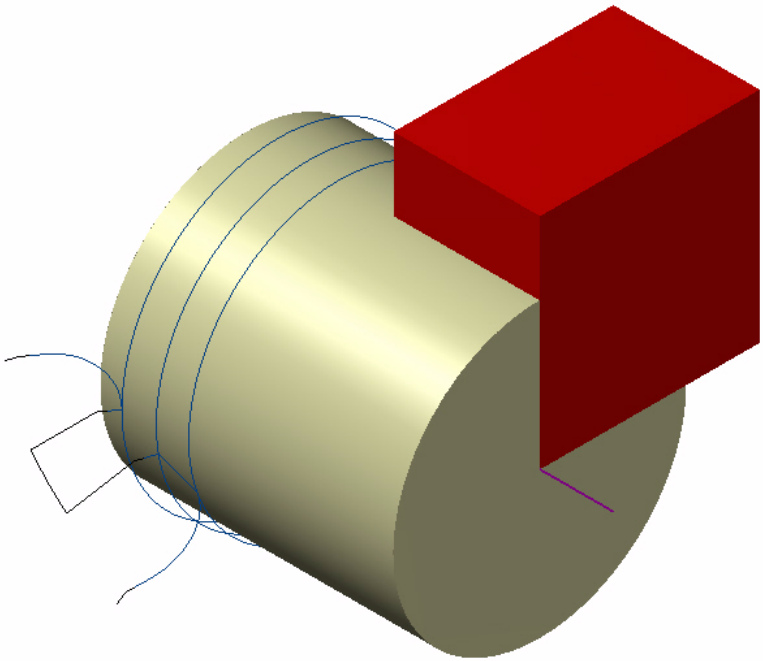
▶ Selecting the gouge-check strategy

1 Open the toolpath parameters for the operation and select the **Gouge check** tab.

2 Select the **Stop tool path calculation** strategy.

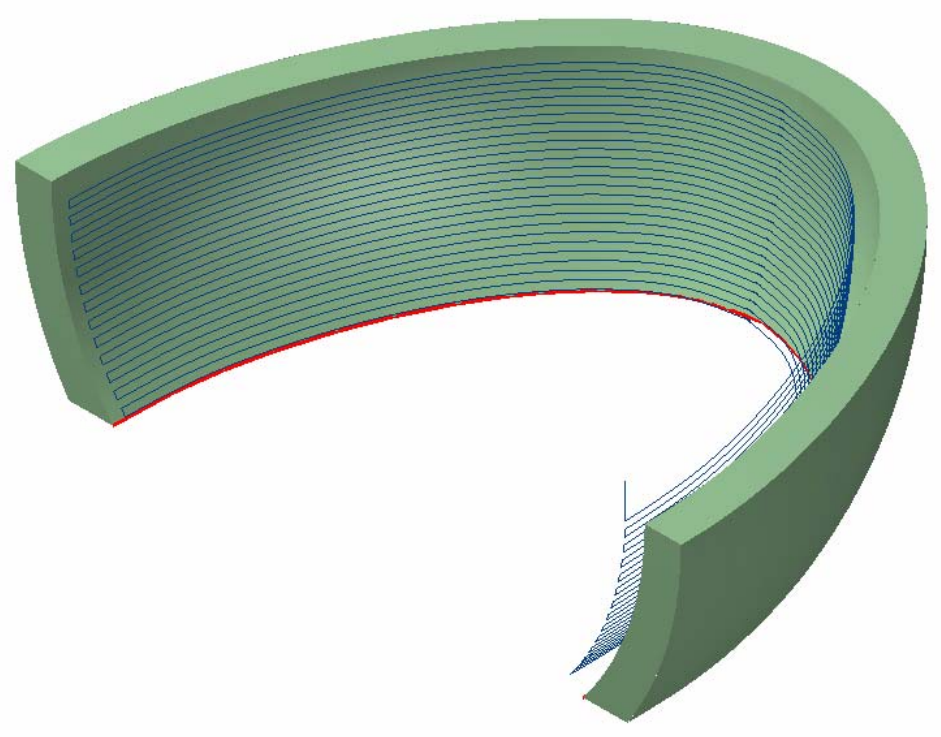


3 Click **OK** and then regenerate the toolpath. Mastercam generates a partial toolpath and stops toolpath calculations when it detects a gouge.



Tutorial 2: Multiaxis Toolpath Techniques

The second tutorial introduces a number of more sophisticated multiaxis techniques to machine the interior of this part. You will use a 10mm flat endmill.



Some of the skills you will learn include:

- creating a swarf toolpath
- handling problem areas in your part model
- creating multiple collision checks for part fixtures and other check surfaces

The part geometry and tool are saved in the following file:

\Documentation\ExampleParts\Multiaxis_Example2.mcx

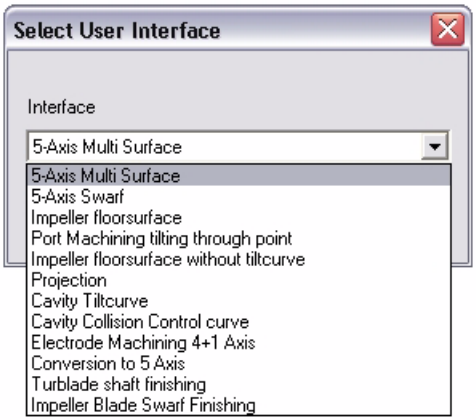
Load this part to begin the tutorial.

Note: All the parts and tools in this tutorial are metric.

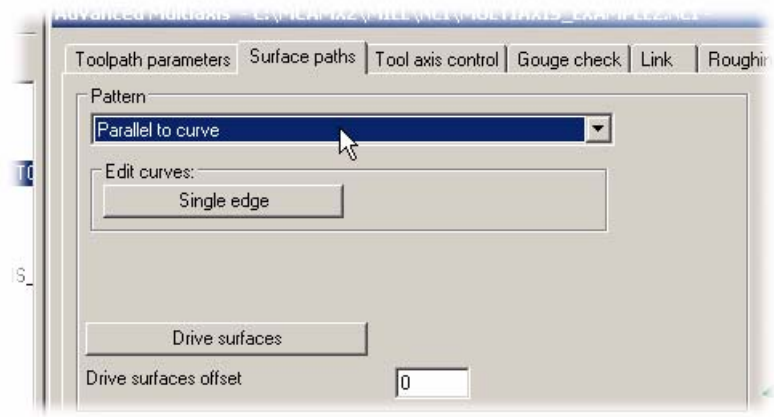
Exercise 1: Creating cuts parallel to a leading curve

▶ Selecting the geometry

1. From the **Toolpaths** menu, choose **Advanced Multiaxis**.
2. Select **5-Axis Multi Surface** and click **OK**.

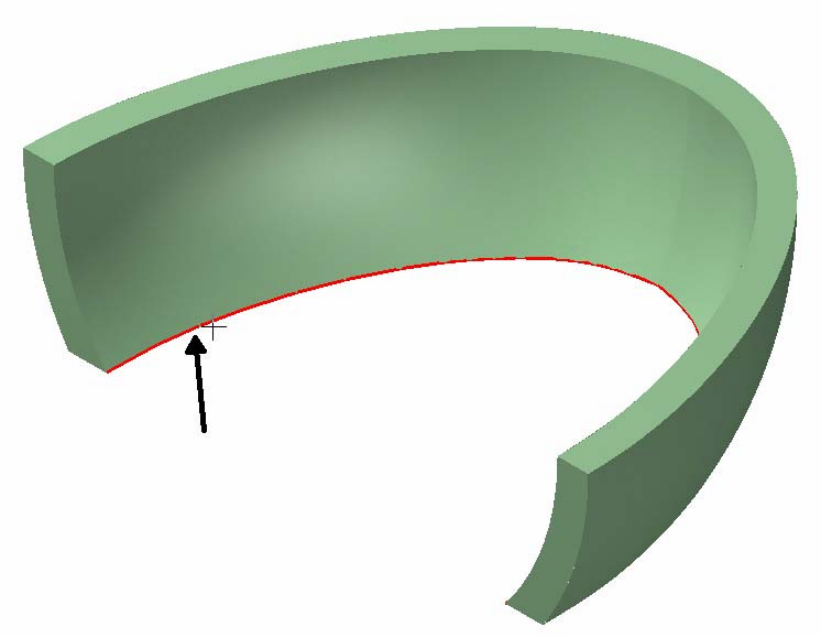


3. On the **Toolpath parameters** tab, select the 10mm flat endmill.
4. Select the **Surface paths** tab.
5. Select **Parallel to curve** as the cutting pattern.

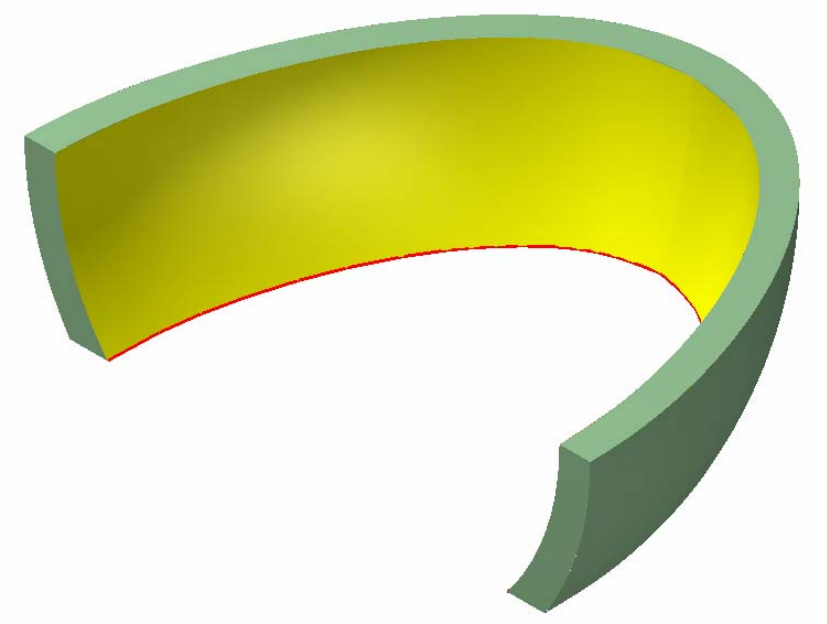


This option aligns the cut direction parallel to a leading curve. Mastercam's advanced multiaxis toolpaths can handle convex surfaces, which are very common in injection molds and forging dies.

6 Click **Single edge** and select the red curve.



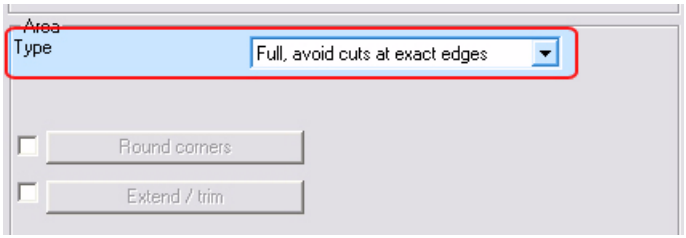
7 Click **Drive surfaces** and select both highlighted surfaces.



8 Click **Done**.

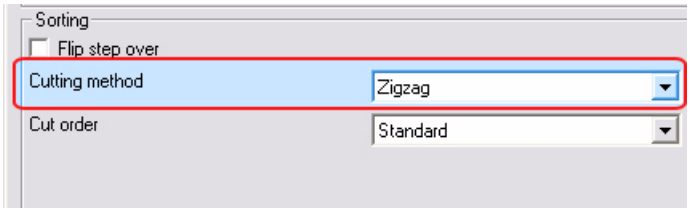
▶ Configuring the cutting passes

1 In the **Area** section, set the type to **Full, avoid cuts at exact edges**.

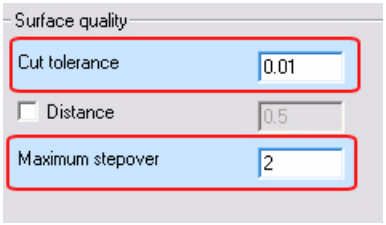


With this setting, the toolpath will be generated on the whole surface, but a cutting pass will not be calculated on the exact edge of the surface. This is useful in case the boundary of the drive surfaces is not smooth. The maximum distance from the edge will be less the half the maximum stepover.

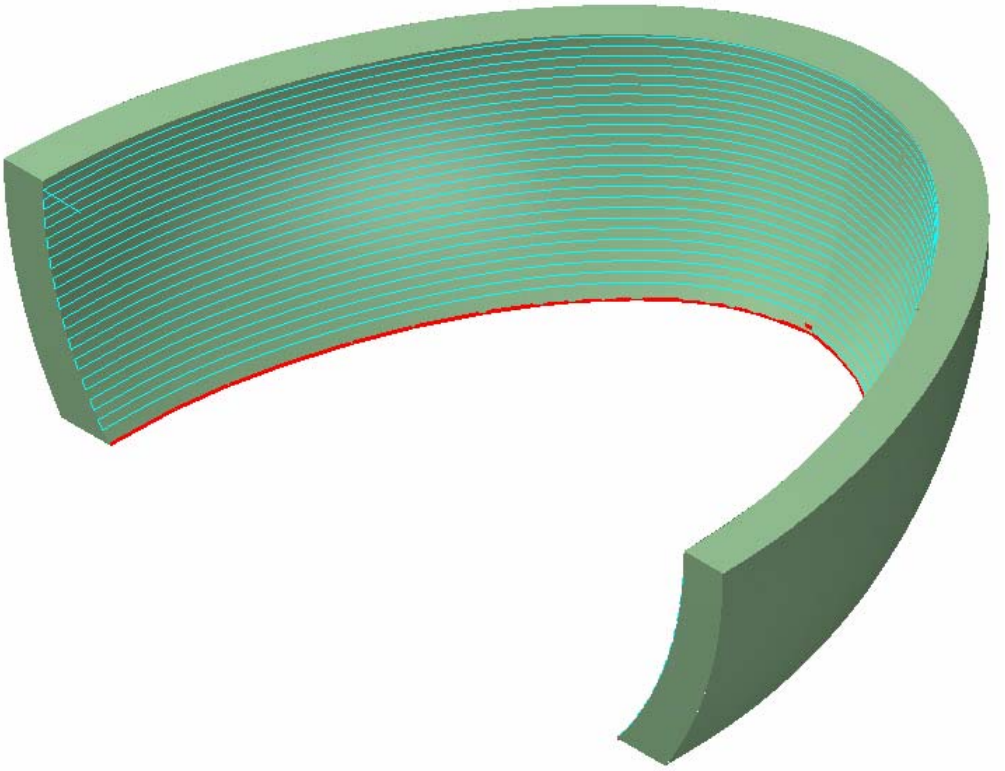
2 Select the **Zigzag** cutting method.



3 Set the **Cut tolerance** to **0.01** and **Maximum stepover** to **2**.

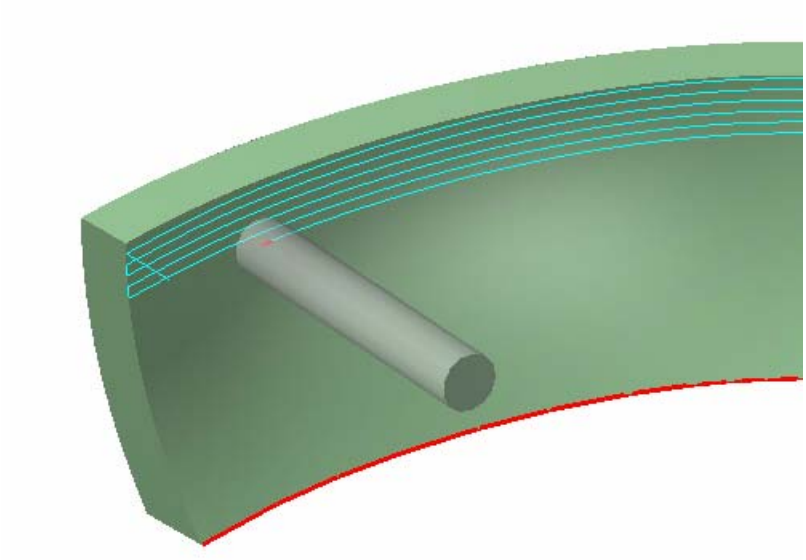


4 Click **OK** to generate the toolpath. It should look like the following picture.



Exercise 2: Creating a swarf toolpath

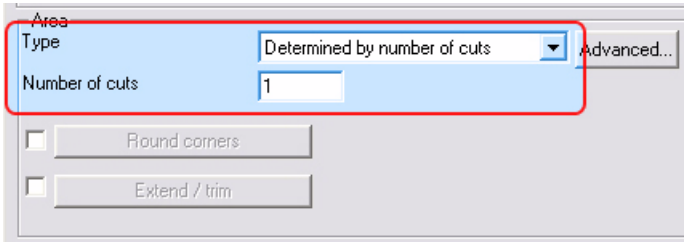
The toolpath in the previous exercise was machined with the tool in an almost horizontal position:



In this exercise, you will modify this toolpath to produce a true swarf toolpath, with a single cut along the bottom rail. This exercise is also a good illustration of how tool axis control and gouge-check settings work together to meet sophisticated tool control needs.

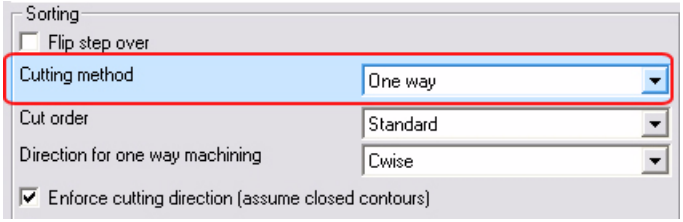
▶ Editing the cutting pattern

1. In the Toolpath Manager, make a copy of the previous operation and click its **Parameters** icon to edit it.
2. Go to the **Surface paths** page.
3. Change the area type to **Determined by number of cuts** and set the **number of cuts** equal to **1**.



Since this will be a swarf toolpath, you only need a single pass along the curve.

4 Set the **Cutting method** to **One way**.

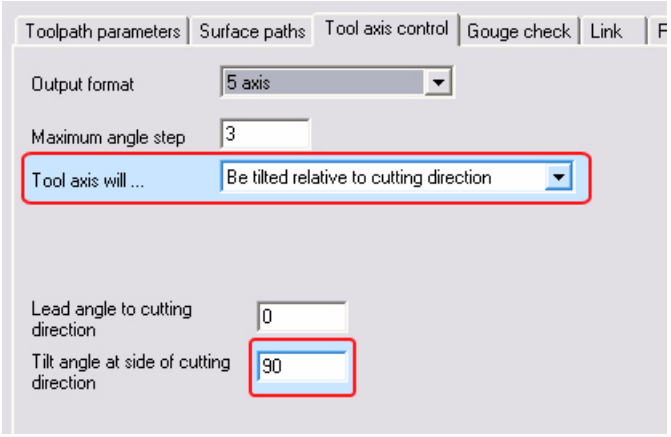


▶ Setting the tool axis control

1 Go to the **Tool axis control** tab.

2 Choose the **Tool axis will be tilted relative to cutting direction** strategy.

3 Set the **Tilt angle at side of cutting direction** to **90**.



The 90 degree side tilt changes the tool from its almost horizontal orientation, normal to the surfaces, to a vertical orientation suitable for swarfing.

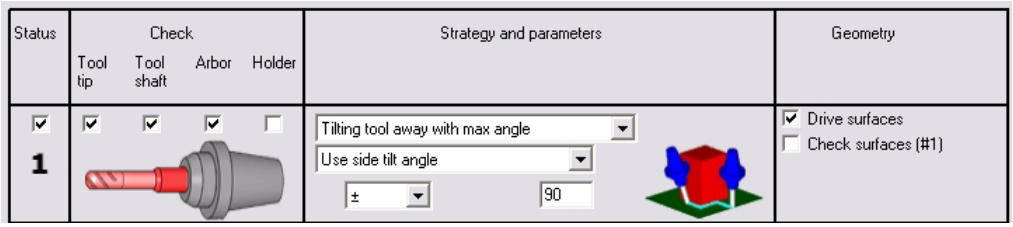
▶ Creating the gouge-check strategy

1 Go to the **Gouge check** tab and activate strategy 1.

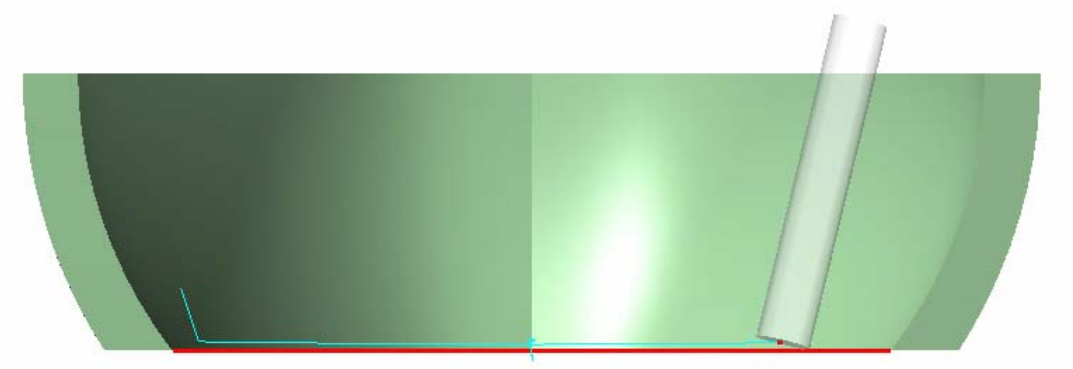
2 Select the **Tool tip, Tool shaft,** and **Arbor** to check against.

3 Select the **Tilting tool away with max angle** strategy, and select **Use side tilt angle** option. This will tilt the tool away from the drive surfaces just enough to avoid gouging.

4 Clear the **Check surfaces** option, since you are only working with drive surfaces in this part. Your settings should look like this:



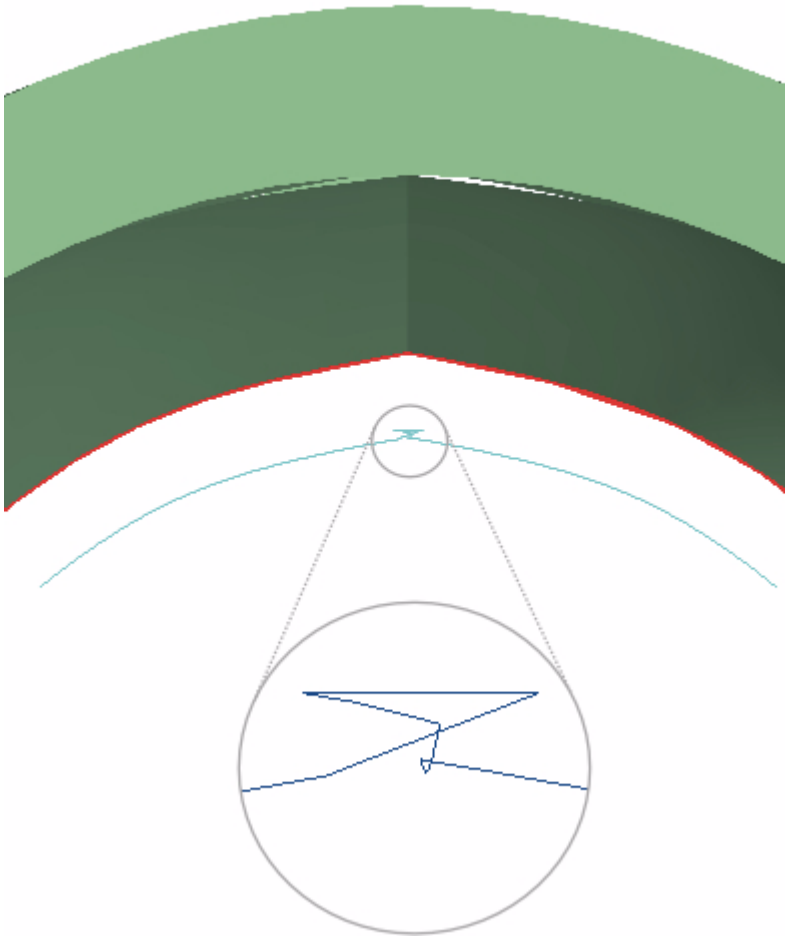
5 Click **OK** and then regenerate the toolpath. You should see a single cut with the tool axis aligned with the drive surfaces.



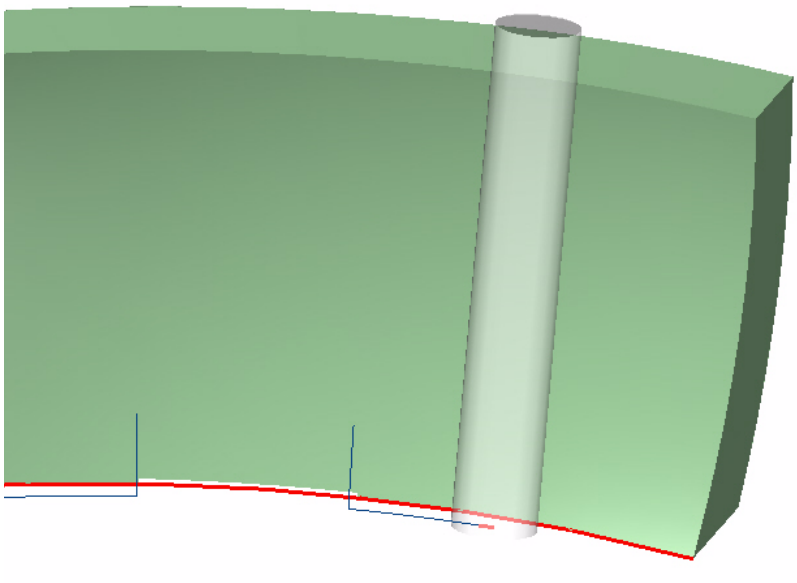
Exercise 3: Handling problem areas in the part model

If you look closely at the toolpath from the previous exercise, you can see several problem areas.

First, there is a sharp corner where the two surfaces meet. This results in a “fishtail” in the toolpath:



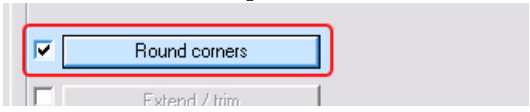
Second, there is a gap in the drive surface that causes the tool to retract, even though the lead curve is continuous:



In this exercise you will learn how to handle both these problems.

▶ Eliminating fishtails in your toolpath

1. Open the toolpath parameters for the operation and select the **Surface paths** tab.
2. Select the **Round corners** option.

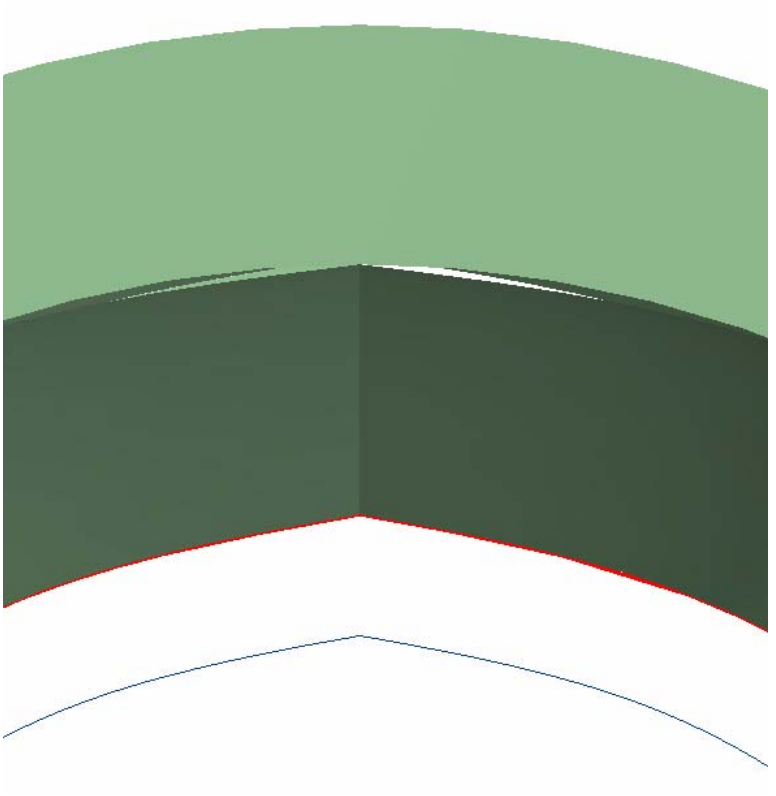


This option works like a fillet generator for your toolpath. The surface model is rounded (filleted) in the direction of the toolpath slices with a radius sufficiently large to smooth sharp corners and corners with very small radii.

The radius used by Mastercam is the radius of the tool plus the current stock to leave value. This option is used most often with ball cutters, lollipop cutters, or conical cutters with a ball tip. Using this option with end mills is also appropriate for swarf machining, like this part.

TIP: Click the **Round corners** button to increase the implied radius by a fixed amount.

3 Click **OK** and then regenerate the toolpath. When you zoom in on the area, you should see a clean transition between surfaces.



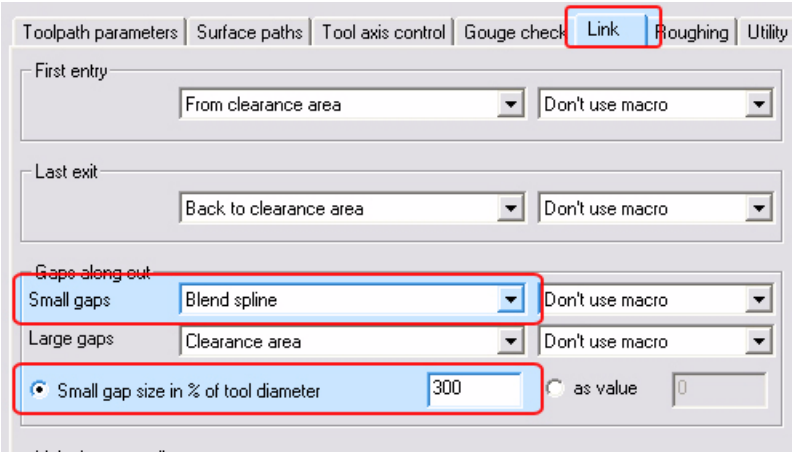
▶ Handling surface gaps in your toolpath

The **Link** tab includes a section that lets you specify how to handle large and small gaps. You can define which gaps are small and large as a percentage of the tool size.

In this case, the gap is approximately 25mm, so you will set the separation between small and large gaps at 300% of the tool diameter, thereby defining this as a small gap.

1 Open the toolpath parameters for the operation and select the **Link** tab.

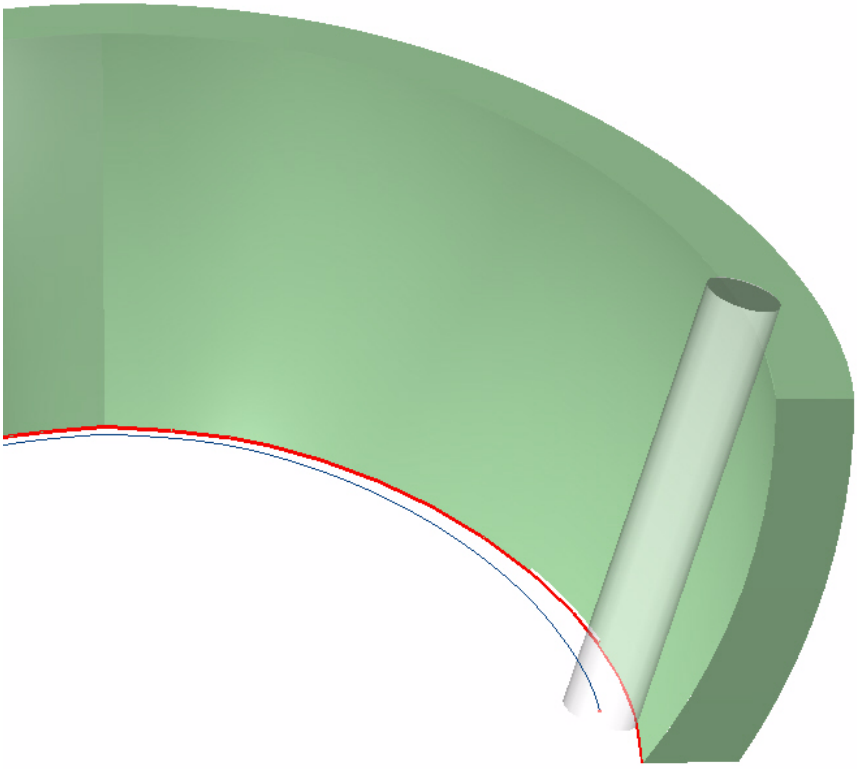
2 Select **Blend spline** as the **Small gaps** strategy.



The goal is to keep the toolpath close to the geometry and not have it retract at all. The blend spline strategy connects the drive surfaces with a toolpath move that leaves and enters the existing surfaces tangentially.

3 Enter **300** to define the **Small gap size**.

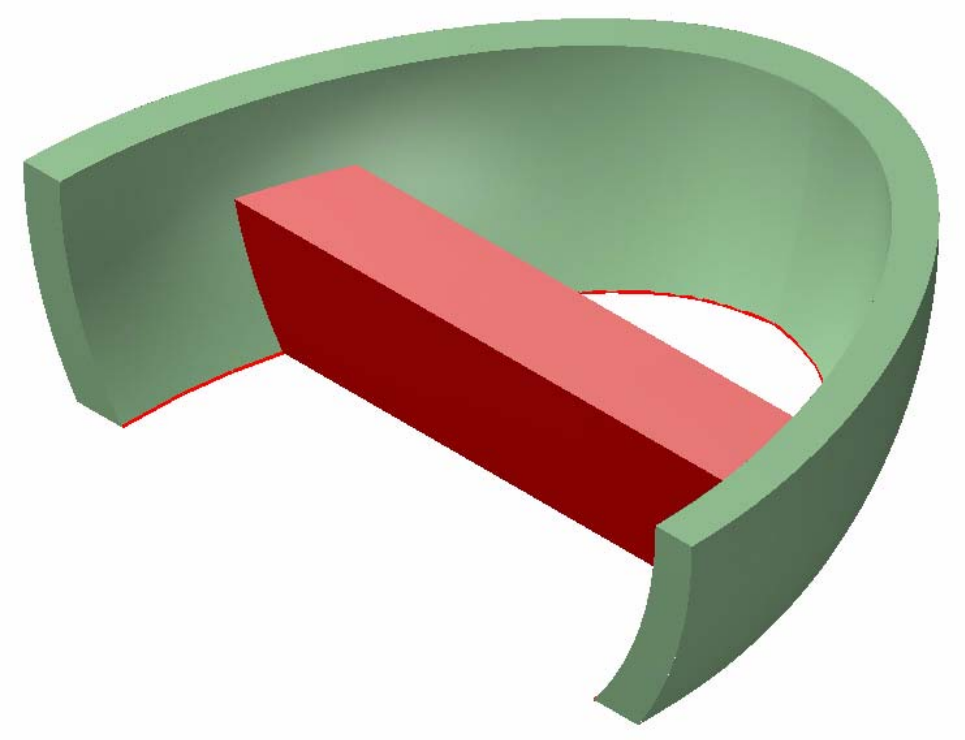
4 Click **OK** and then regenerate the toolpath. When you zoom in on the area, you should see a smooth toolpath in the gap area.



Exercise 4: Combining several gouge and collision checks

In the previous exercises, you used the gouge-checking function to help define the proper tool position for a swarf toolpath. In this exercise, you will apply additional gouge checks to keep the tool away from check surfaces.

Before beginning this exercise, use the Level Manager to make level 100 visible. Your part should now display a horizontal bar:

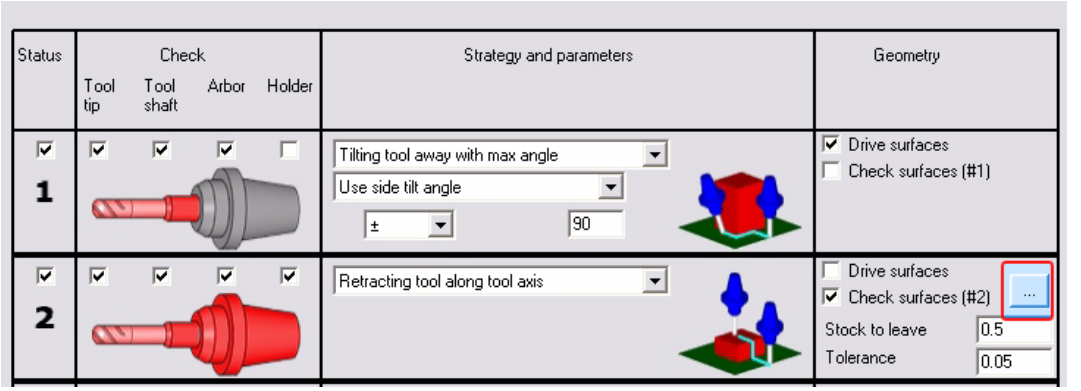


▶ Collision-checking the bar

1. In the Toolpath Manager, make a copy of the previous operation and click its **Parameters** icon to edit it.
2. Go to the **Gouge check** tab.
3. Select gouge check **2** in the **Status** column.
4. Select the **Tool tip, Tool shaft, Arbor** and **Holder** to check against.
5. Set strategy to **Retracting tool along tool axis**.
6. De-select **Drive surfaces** and select **Check surfaces**. There is no need to check the drive surfaces with this test, since they are handled in the first test. For this test, Mastercam will only look at the check surfaces.
7. Enter **0.5** as **Stock to leave**, and **0.05** as the **Tolerance**.

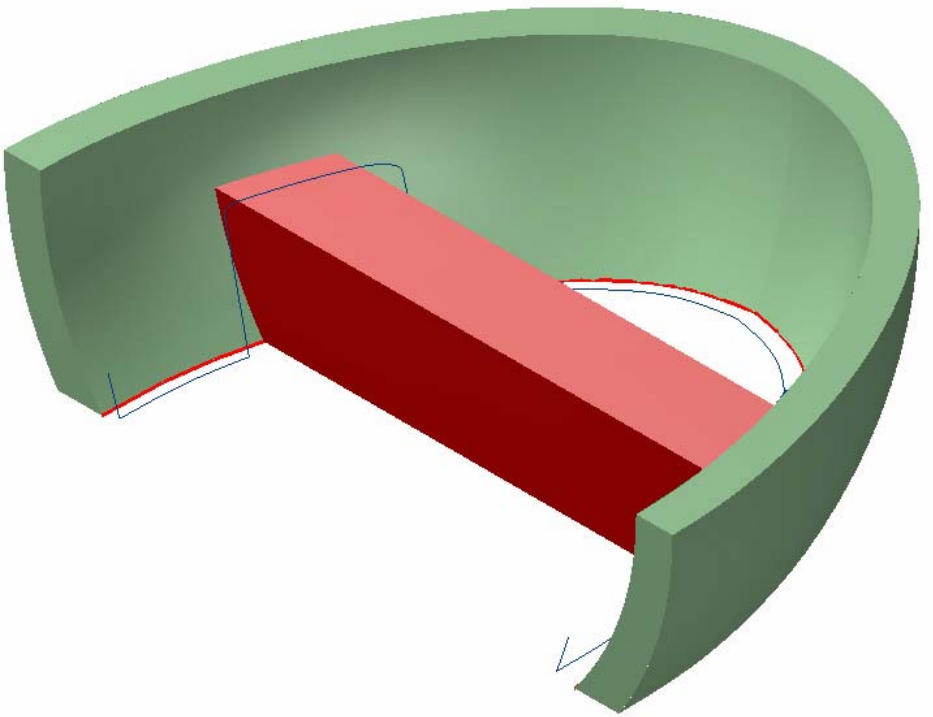
The goal of this setting is to prevent machining the check surfaces, while staying within 0.5 mm of them.

Your settings should look like this:



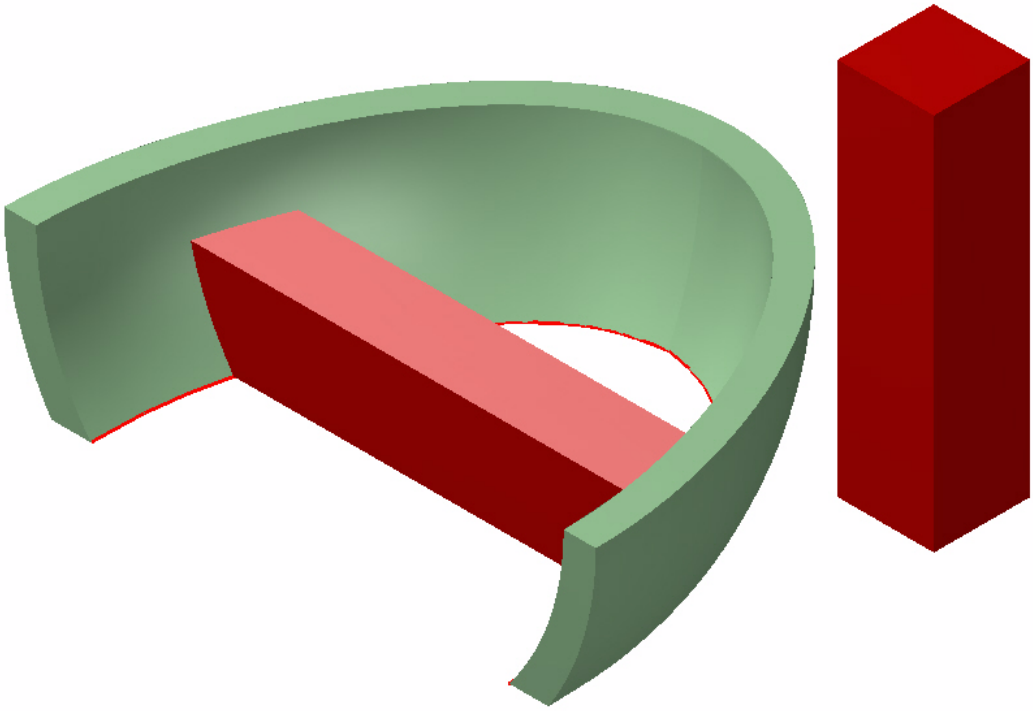
8 Click the **[…]** button and select all the red surfaces.

9 Click **OK** and then regenerate the toolpath. The toolpath should clear the bar with a 0.5mm clearance like in the following picture.



▶ Collision-checking external fixtures

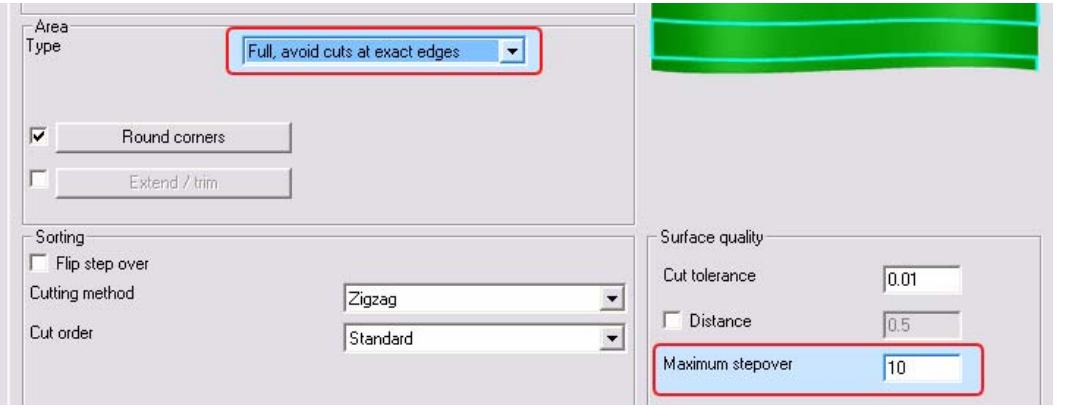
You will now modify the collision checks to verify that the tool and holder do not collide with an external fixture. Before continuing, make level 101 visible in your part file to expose these surfaces.



You will also change the cut pattern, to create additional cutting passes near the top of the surfaces to check against the fixture.

1. Open the toolpath parameters for the operation and select the **Surface paths** tab.
2. Change the **Area** type to **Full, avoid cuts at exact edges**.
3. Change the **Cutting method** to **Zigzag**.

4 Change the **Maximum stepover** to **10**.



5 Go to the **Gouge check** tab

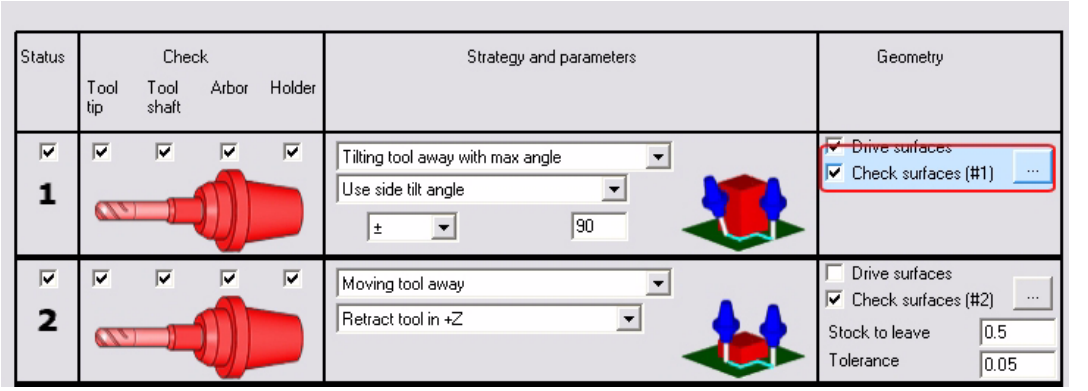
6 Edit gouge check 1:

- Add the **Holder** to check against. (See the picture below.)
- Select **Check surfaces** and select the surfaces for the red fixture.

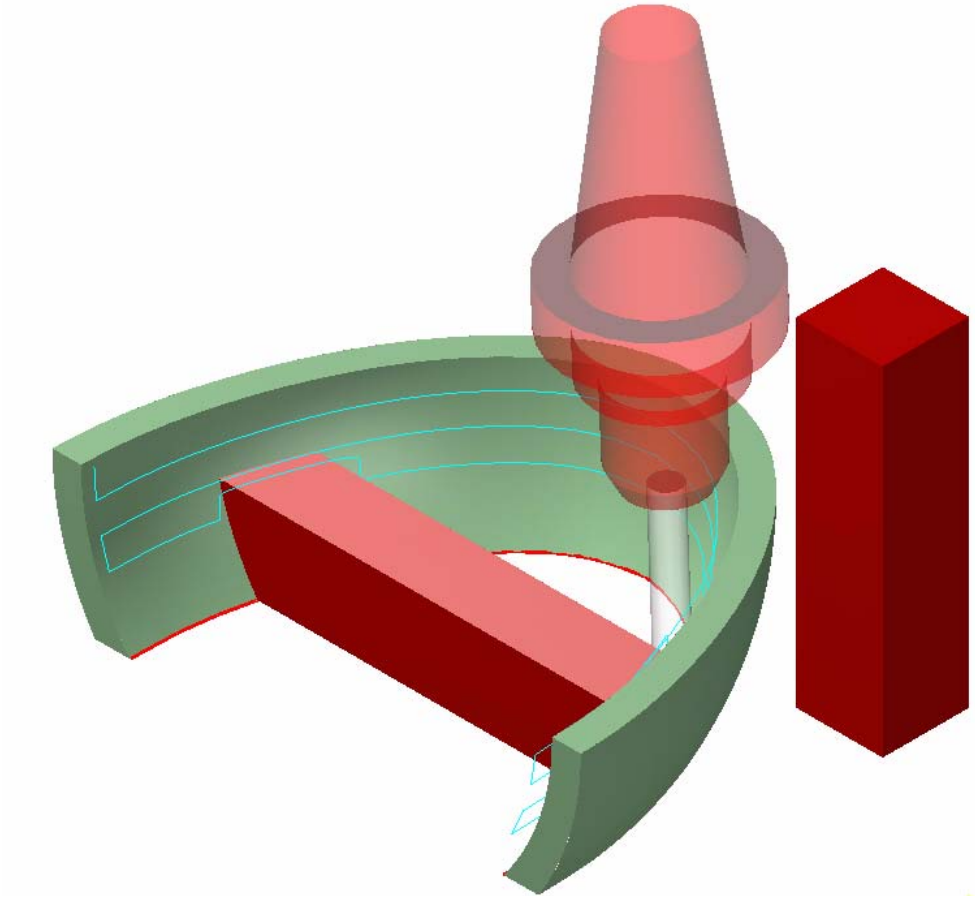
7 Edit gouge check 2:

- Select the **Moving tool away** strategy.
- Set the retract option to **Retract tool in +Z**.

Your settings should look like the following picture:

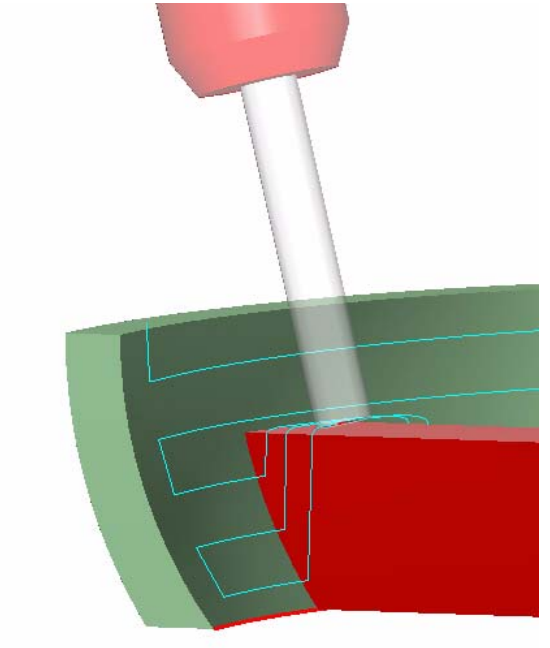


8 Click **OK** and then regenerate the toolpath. You can see the tool axis tilt when the holder approaches the fixture.



Also, look at the new retract strategy around the bar. The tool retracts straight up in Z instead of along the tool axis, which is

a much more appropriate strategy with the multiple cutting passes.



Congratulations! You have completed the tutorial portion of this book. Continue with the next chapter to learn about specific application examples and how they are created.

chapter 2

Application Examples

This chapter introduces several application examples. Instead of giving you step-by-step procedures like the tutorials in the previous chapter, this chapter discusses how to apply the advanced multiaxis toolpaths to specific types of parts and explains the key settings that make each toolpath appropriate for its application.

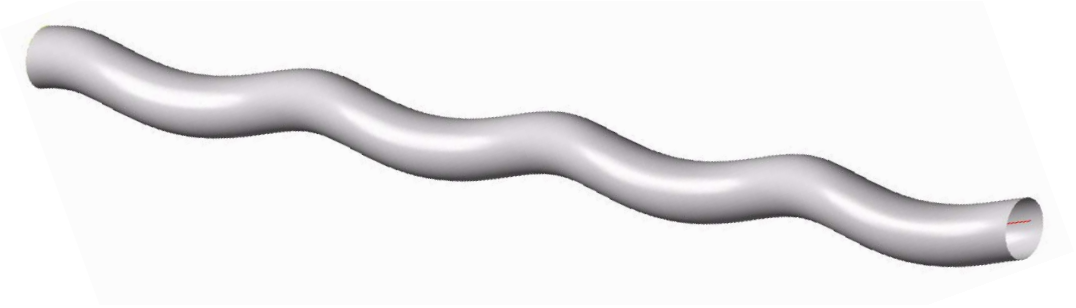
It includes the following examples:

- **Example 1:Compressor Armature with 4-Axis Output** *page 58*
- **Example 2:Engraving** . . . . . *page 63*
- **Example 3:Camshaft on a Multitasking Lathe** *page 71*
- **Example 4:Electrode Machining**. . . . . *page 79*
- **Example 5:Impeller Floor Roughing** . . . . . *page 91*
- **Example 6:Impeller Blade Semi-finishing**. . . . *page 101*
- **Example 7:Impeller Floor Finishing** . . . . . *page 107*
- **Example 8:Impeller Blade Finishing** . . . . . *page 113*

Completed parts for each exercise can be found in the **\Documentation\ExampleParts** folder in your Mastercam installation folder. (The four impeller examples all use the same part.) Remember that for almost any multiaxis application, there are many possible machining strategies. Feel free to use these examples as starting points, trying out different settings and techniques that are relevant to your own work and your own preferences.

Example 1: Compressor Armature with 4-Axis Output

In this example, you will machine the outside of this compressor armature. The part lies along the X axis.

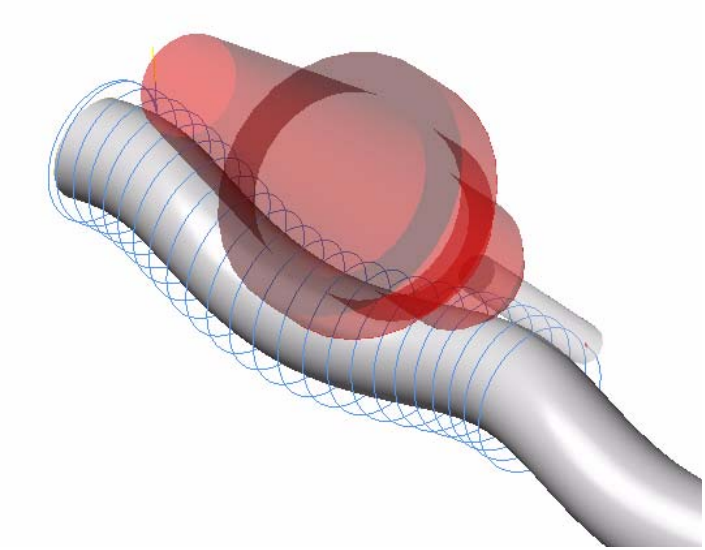


The approach will be to machine with the side of the tool, using a 4-axis toolpath that rotates around the part in a continuous spiral. You can typically get much better surface finishes, longer tool life and shorter cycle times when cutting with the side instead of the bottom of the cutter, so you will use the advanced multiaxis toolpaths to take advantage of this.

To begin, open the following file:

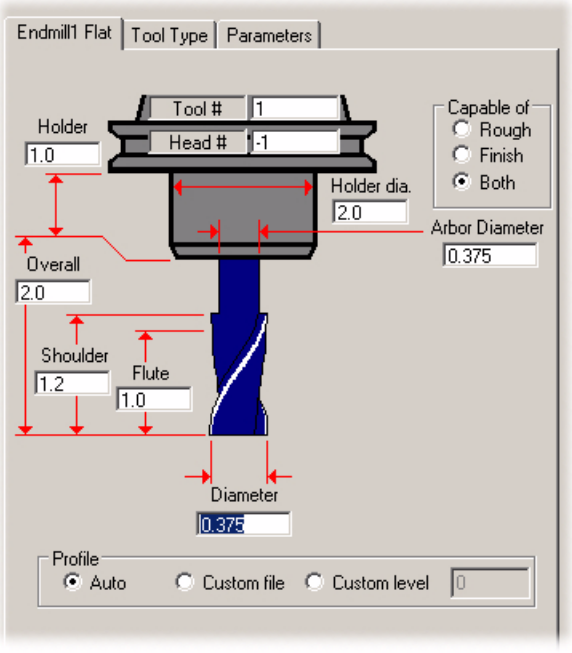
\Documentation\ExampleParts\Compressor_armature.mcx

The following picture shows the desired cutting motion:



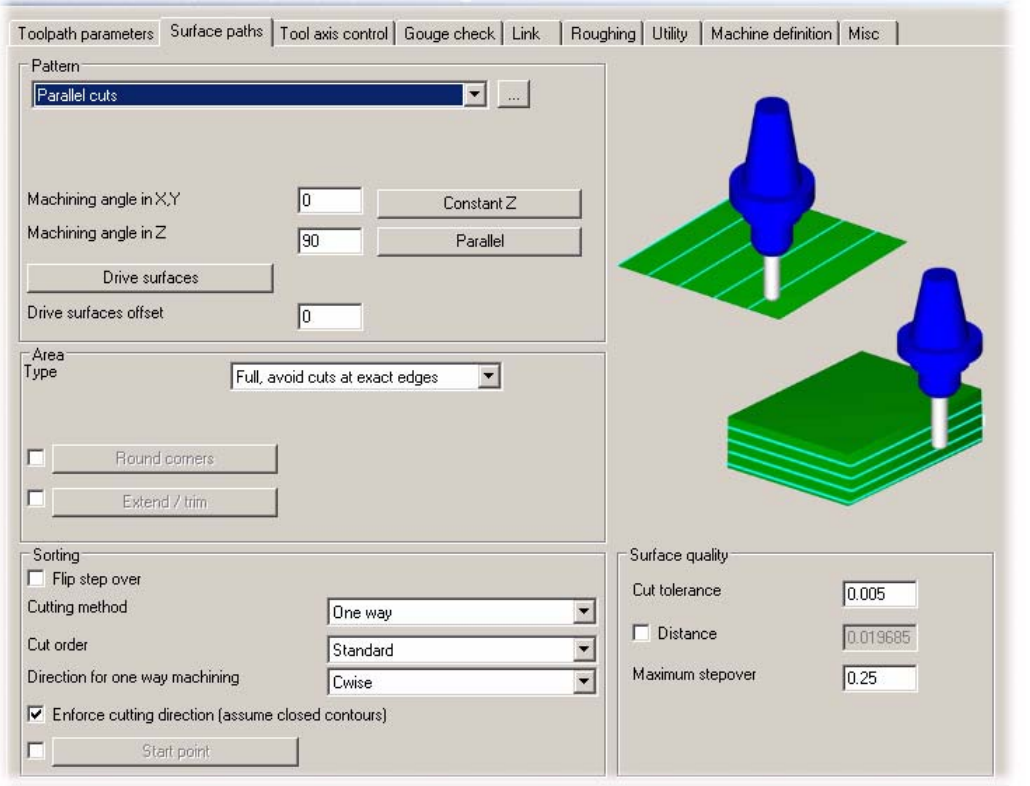
Tooling

For this part, use a 0.375-inch flat endmill.



Drive geometry and cutting passes

Use the following **Surface paths** settings:



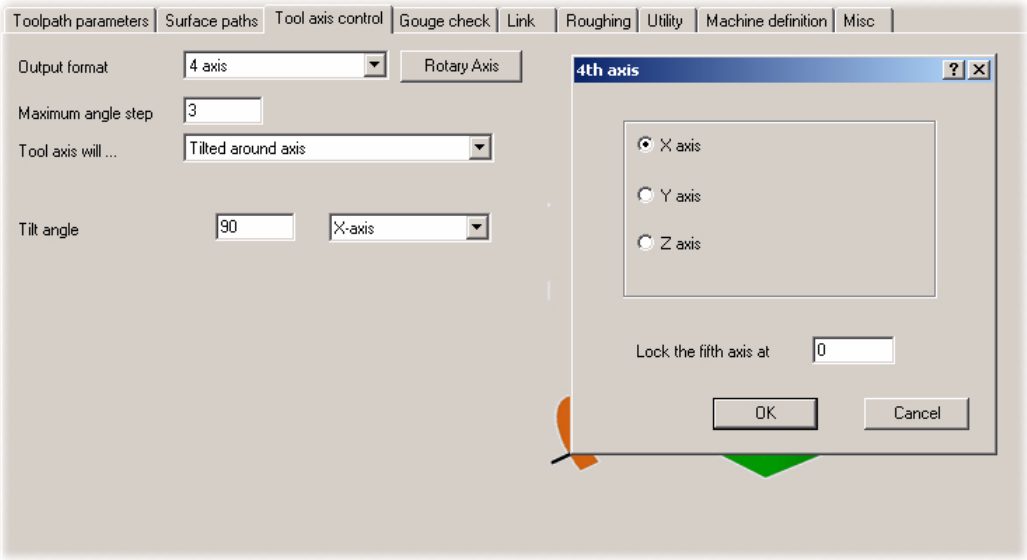
- Use the **Parallel cuts** pattern to create the cutting passes.
- Each cut needs to be parallel to a vertical plane, so set the **Machining angle in Z** to **90**. Since the part is parallel to the X axis, the **Machining angle in X,Y** should be left at **0**.
- For **Drive surfaces**, select the single armature surface.
- Select the **One way** cutting method to keep rotating the part in the same direction for all the passes.

In addition, select the following option on the **Utility** tab:

This converts all of the separate parallel cuts into a single continuous spiral, which is ideal for this part.

Tool axis control and gouge checking

Use the following **Tool axis control** settings:



- To generate the 4-axis output, simply select it as the **Output format** and click the **Rotary Axis** button to select the desired axis.
- The other settings lock the tool at a 90-degree angle to the X axis. As the part rotates, the side of the tool will machine it.

There is no need for gouge checking for this toolpath, so all the options on the **Gouge check** tab are turned off.

In the next example, you will learn about settingup an engraving application.

Example 2: Engraving

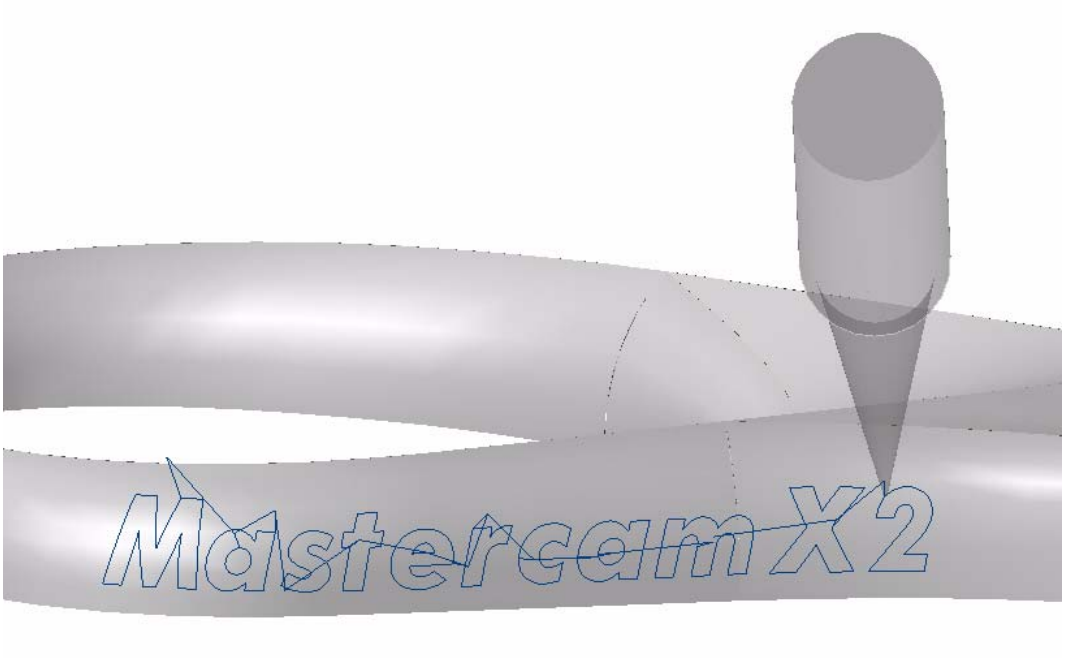
5-axis engraving is popular in the mold and die industry. There are company logos, recycling icons, volume markers, and artistic designs engraved on molding surfaces deep inside cavities or on free-form surfaces on cores. In this example, you will engrave the words **Mastercam X2** on the side of the part as shown below:

You must use a small cutter, and you can only get superior results if you stay normal to the surfaces you are cutting—5-axis motion is a requirement. In this example, you will use a slender V-groove cutter with a 3-degree taper. The 5-axis engraving toolpath maintains the depth of the cut contour relative to the drive surface. This is especially noticeable when using V-groove cutters, since the width of the engraved contour will remain constant.

To begin, open the following file:

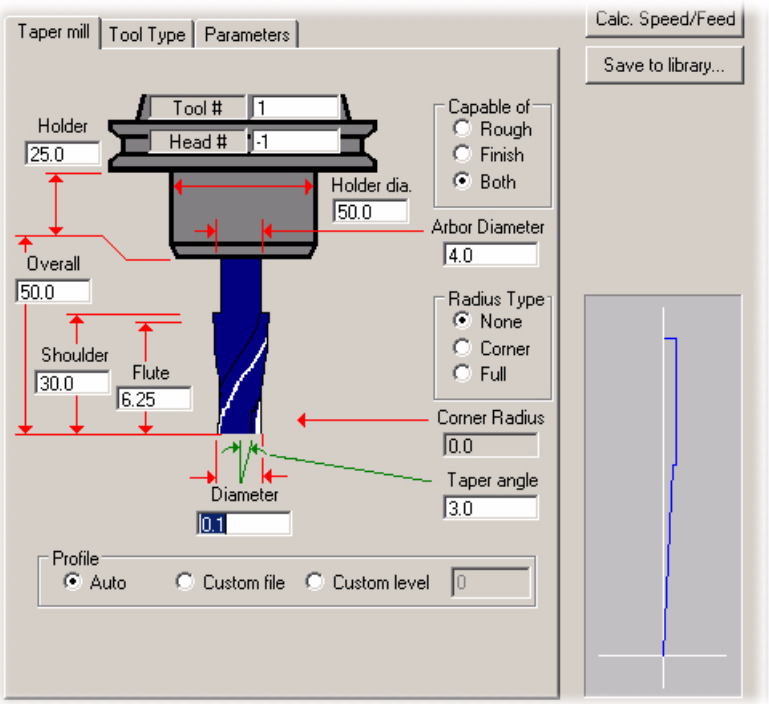
\Documentation\ExampleParts\Engraving.mcx

The following picture shows the desired toolpath:



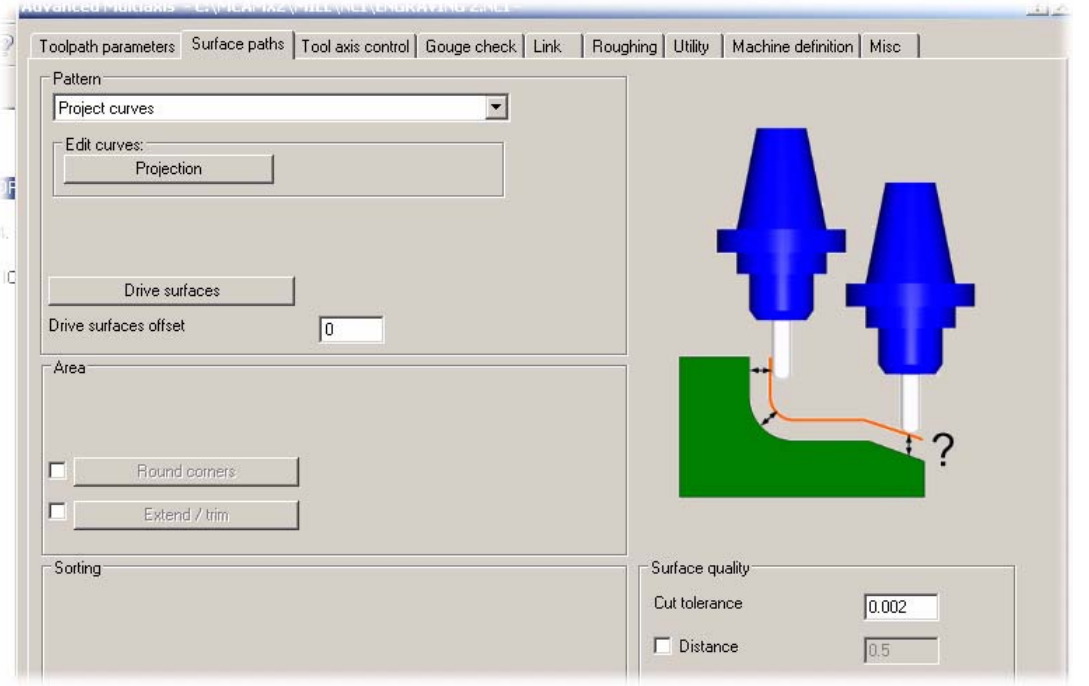
Tooling

For this part, use a tapered endmill with the following dimensions. Note that this is a metric part so these dimensions are in mm.



Drive geometry and cutting passes

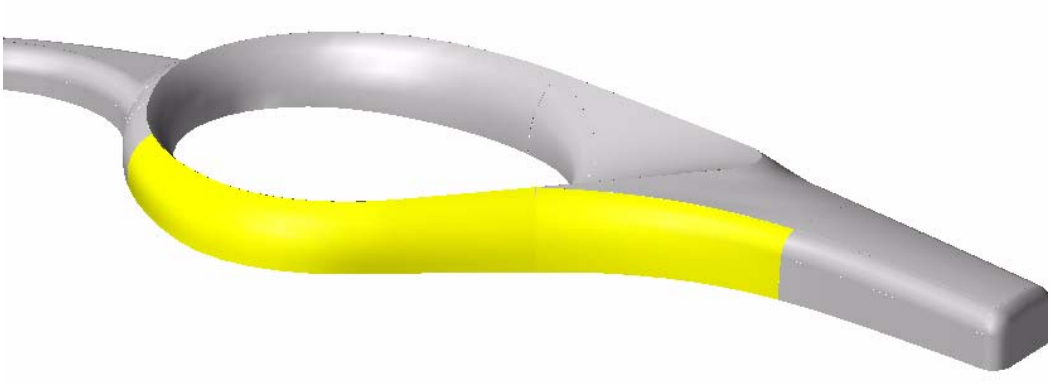
Use the following **Surface paths** settings:



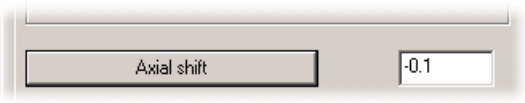
- Use the **Project curves** pattern to create the cutting passes.
- Click the **Projection** button and select the geometry to be engraved. (In the sample file, the letters are organized by themselves on level 20 to make selection easier. You could also select them easily using a color mask.)

IMPORTANT: The curves must already lie on the surfaces before machining.

- For drive surfaces, select just the surfaces shown:

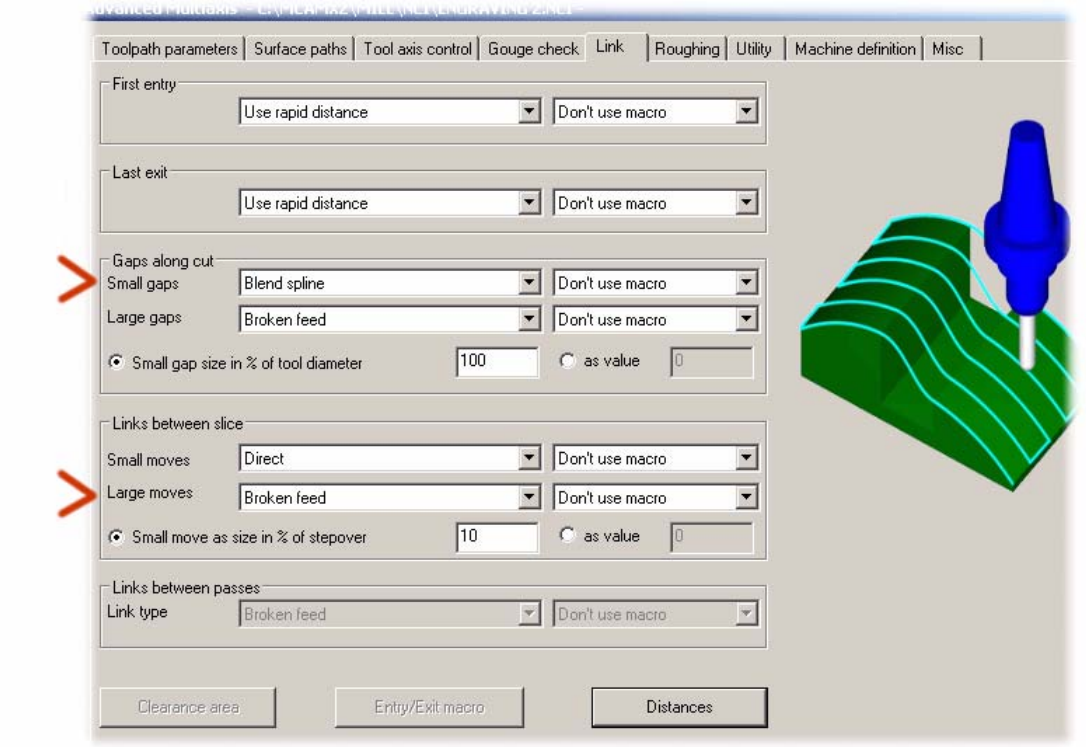


- To control the engraving depth, use the **Axial shift** parameter on the **Utility** tab. This setting shifts the toolpath along the tool axis by a specified amount. Use a negative amount to cut below the surface. In this case, the value of –0.1mm means that the tool will cut just below the surface.



Linking the cuts

Use the following settings on the **Link** tab:

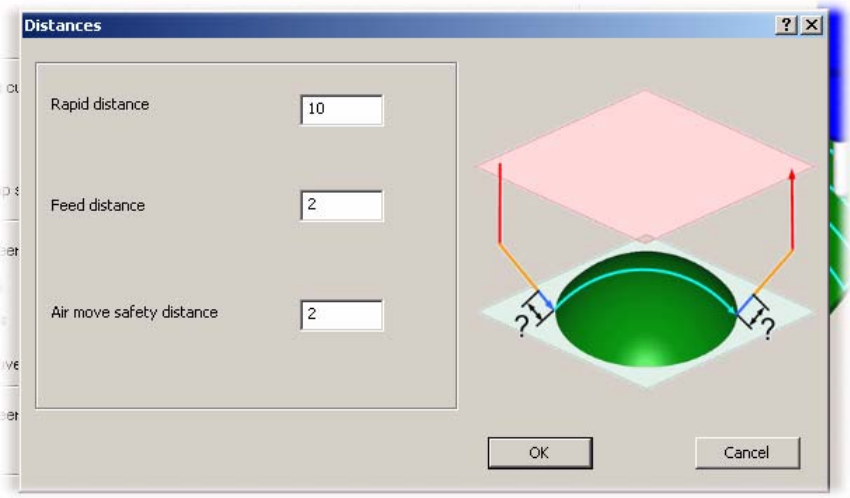


The two settings that are marked in the picture are the key ones.

- Use the **Gaps along cut** settings to control what happens if there are small gaps or discontinuities in the geometry within each letter. In this case, select the **Blend spline** strategy to have the tool continue to feed along a smooth, tangential connecting spline between any gaps. Set a threshold of **100%** of the tool diameter to define when a gap is considered small. For gaps larger than this, the tool will retract from the toolpath.
- Use the **Links between slices** section to control how the tool moves between letters. In this case, **Broken feed** means that the tool will continue at the feed rate, but will retract to a feed plane to move to the next letter. The size threshold is, intentionally, very small—10% of the tool diameter. This is because you don't know the exact spacing between letters, so the amount is set very low to capture all of the possible

transitions. (The dialog box says **% of stepover**, but since there is no stepover for a Project toolpath, Mastercam uses the tool diameter instead.)

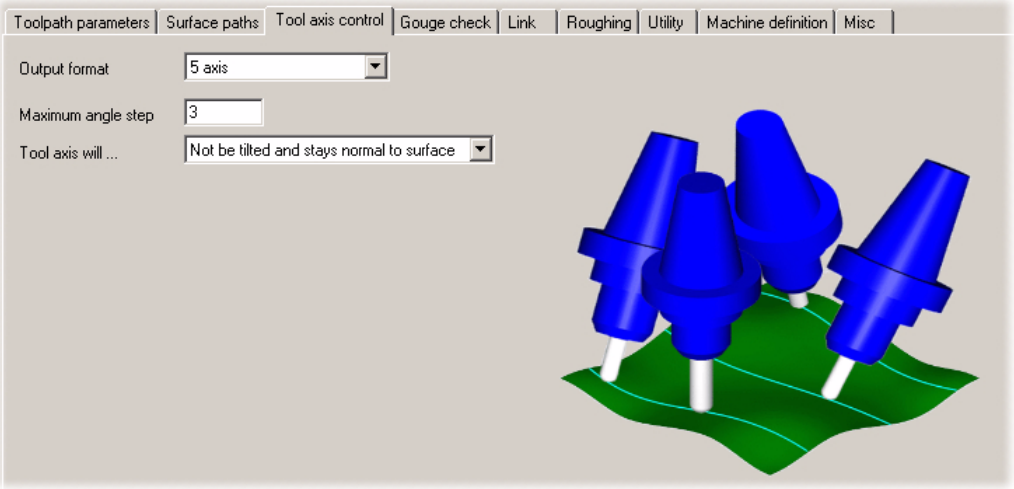
The retract distance is set by clicking the **Distances** button.



In this case, the **Feed distance** is set at 2mm, so the tool will retract 2mm from the surface between each letter.

Tool axis control and gouge checking

Use the following **Tool axis control** settings:



For this application, it is important that the tool stay normal to the surface and not otherwise tilt. The 5-axis output is required to maintain this orientation.

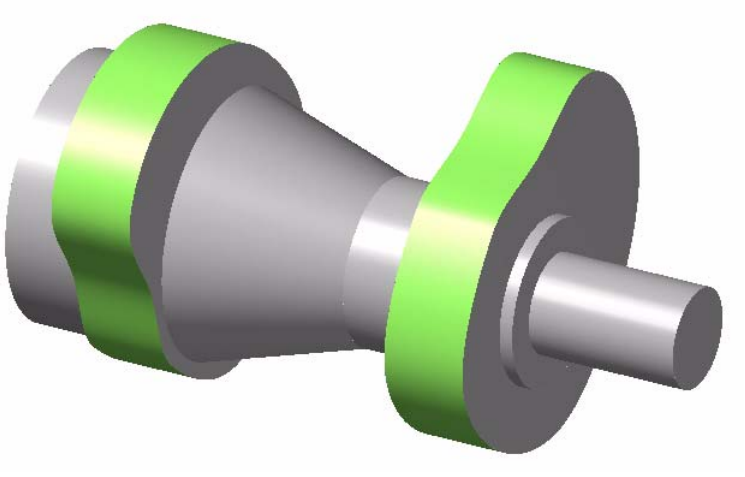
Gouge checking is turned off for this toolpath because it is not needed.

The next example introduces a new type of application, where the advanced multitasking toolpaths are used on a multitasking lathe.

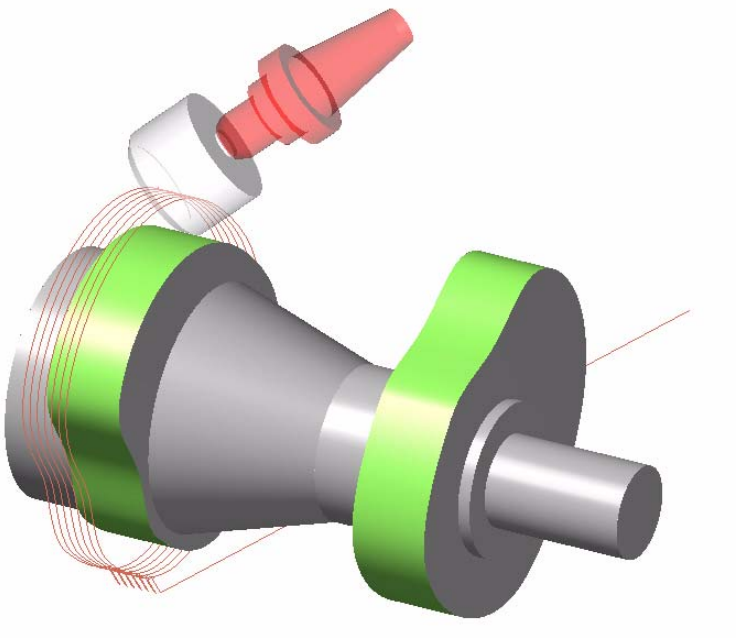
Example 3: Camshaft on a Multitasking Lathe

Multitasking lathes or mill/turn machines are blurring the line between milling and turning. They are becoming more affordable and accurate and let you machine an extremely wide variety of parts. You can complete a part in one setup faster than ever before, but the programing demands are complex. Traditional axis substitution is sometimes very limited especially if you like to use advanced cutting techniques. In many cases, if you want to machine complex contours with bigger tools, substituting the Y axis with a rotational move will produce excessive strain on the contour.

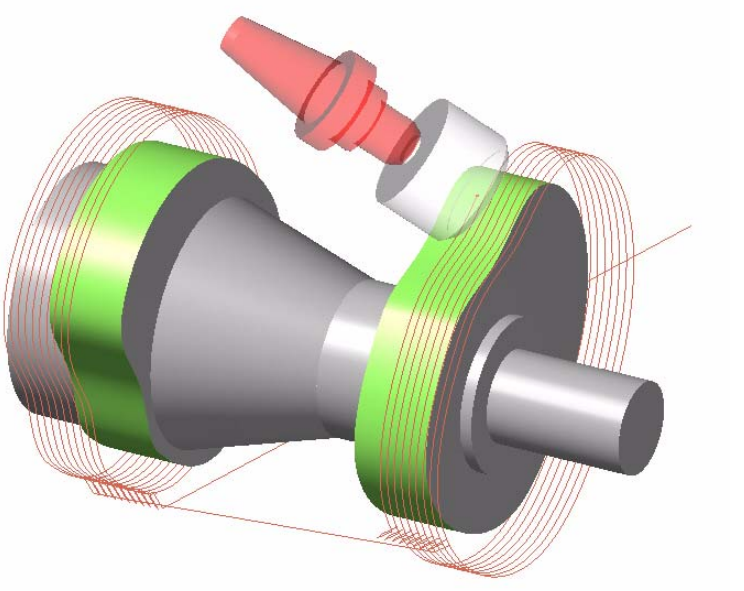
This example demonstrates how to use the advanced multiaxis toolpaths to machine the two cams shown in green in the following picture.



You will machine the contours with the chamfered edge of a large endmill as shown in the following picture. The tool is mounted in the B-axis spindle of a mill/turn lathe.



To machine the other cam, create a second toolpath with the tool rotated in the opposite direction:

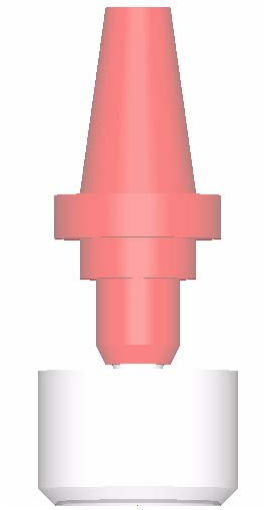


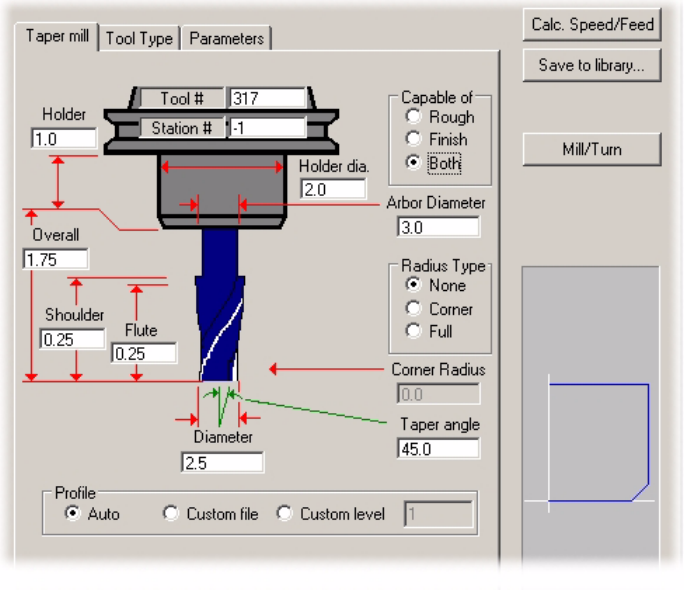
To begin, open the following file:

\Documentation\ExampleParts\Camshaft.mcx

Tooling

For this part, use a tapered endmill with the following dimensions.

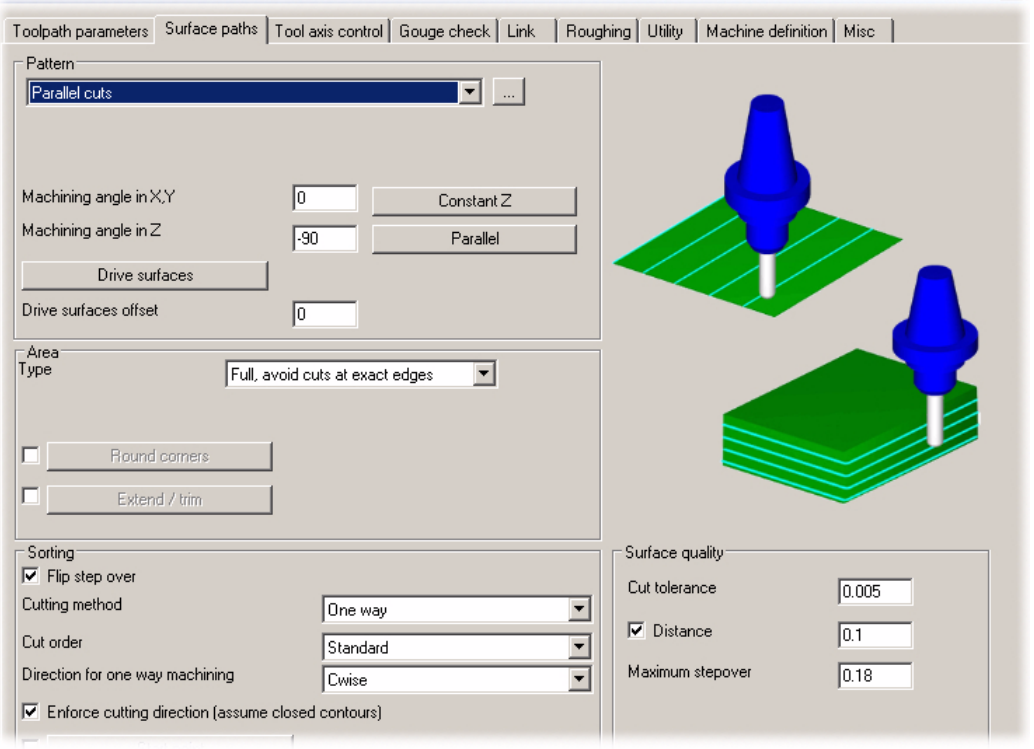




The advanced multiaxis toolpaths do not support the **Face mill** or **Chamfer mill** tool types, so this tool is defined as a **Taper mill**.

Drive geometry and cutting passes

For this part, you will create two toolpaths. For the first toolpath, use the following **Surface paths** settings:



- Use the **Parallel cuts** pattern to create the multiple parallel cutting passes around each cam contour.
- Set the **Machining angle in Z** to **–90** to create cuts aligned in a vertical plane.

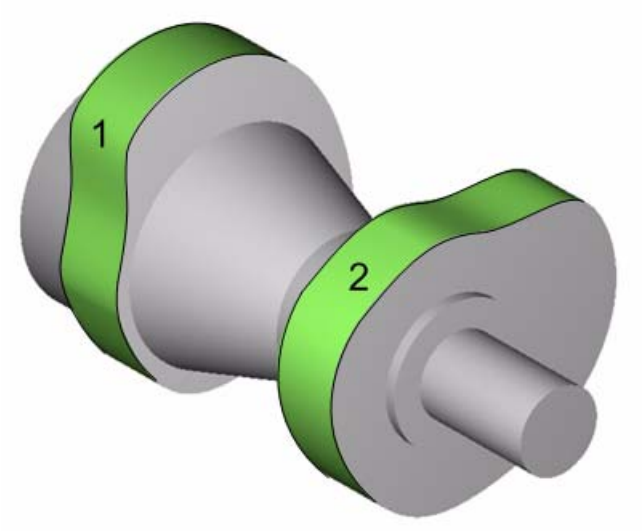
  An angle of **+90** would also align the cuts vertically, but the cutting direction would be reversed. So should you choose **+90** or **–90**? You can verify whether the positive or negative

angle is correct by verifying the cuts against the selected cut **Direction**:

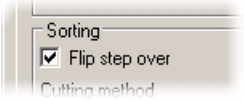
If you imagine that you are in the middle of the lathe, looking towards the left spindle, the tool is properly moving clockwise with the machining angle set to –90. If you select **+90**, then the **Cwise** cut direction would actually produce counterclockwise tool motion.

Note: Even though you're working on a lathe machine, this toolpath uses world (mill) coordinates, with Z pointing up, and the X axis is parallel to the main lathe spindle.

- Click **Drive surfaces** and select Surface 1 to machine just the left cam.



- Select the **Flip step over** option. This reverses the order of the cuts so that as the tool feeds between cuts, more of the cutting force is on the side of the tool, not the bottom.



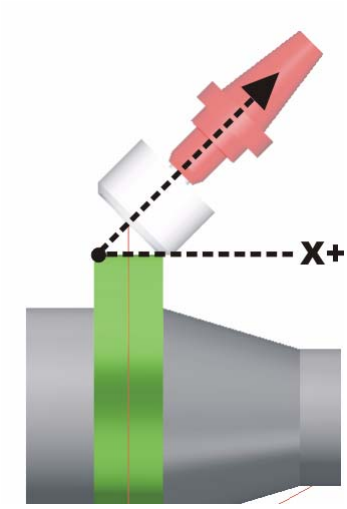
- Activate the **Distance** option. When this option is selected, Mastercam is forced to calculate a tool position before moving the specified distance. It functions like a point generator, ensuring that there is at least one calculated tool position every 0.1 inch.

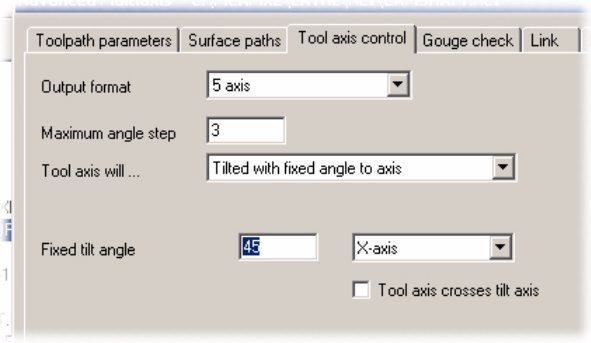
The settings for the second toolpath are exactly the same as the first one, with two exceptions:

- For the **Drive surfaces**, select Surface 2 from the above picture.
- De-select the **Flip step over** option. This reverses the order of the cuts from the first toolpath, which you should do since the tool will be flipped over.

Tool axis control and gouge checking

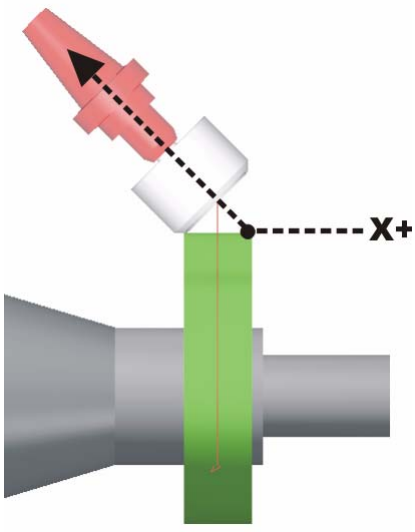
For the first toolpath, use the following **Tool axis control** settings:

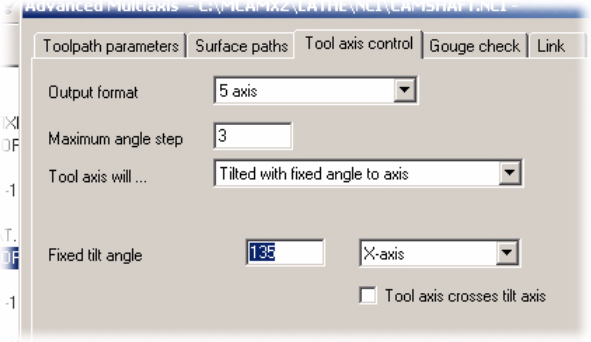




These settings will lock the tool axis at a **45**-degree angle relative to the X axis.

For the second toolpath, flip the tool axis by setting the tilt angle to **135**:



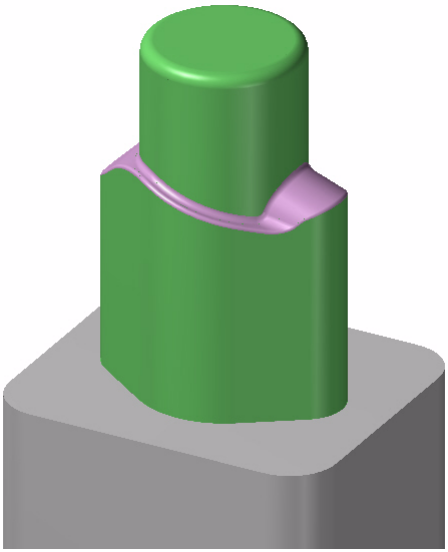


Gouge checking is turned off for this toolpath because it is not needed.

The next example introduces a new application, machining an electrode with a 4+1 axis toolpath.

Example 4: Electrode Machining

Electrodes used for EDM (electrical discharge machining) applications are mirror images of the features which need to be removed from molds and dies. The EDM process is typically one of the final processes of tool making—the parts have been already hardened, ground and only need some finishing touches in hard to reach areas. The quality of the surface finish on the electrode will directly affect the resulting finish on the workpiece. In this example, you will machine the electrode shown in the following picture.

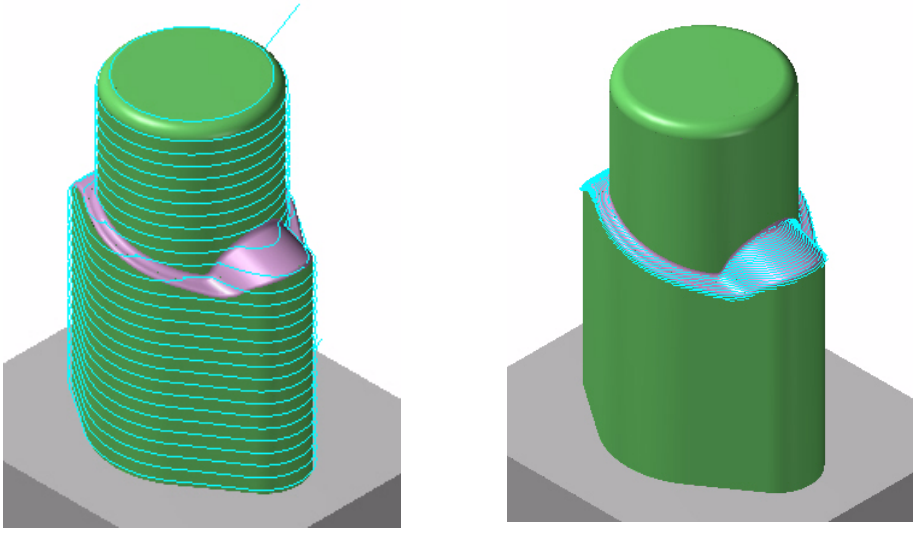


Electrodes like this are typically machined with miniature, tapered ball nose cutters. The best possible surface finish and detail will be achieved with the shortest tool possible. A common practice—followed in this example— is to tilt the tool axis to a fixed position (often by using an aggregate machining head) and then create a toolpath for a 4-axis machine.

To begin, open the following file:

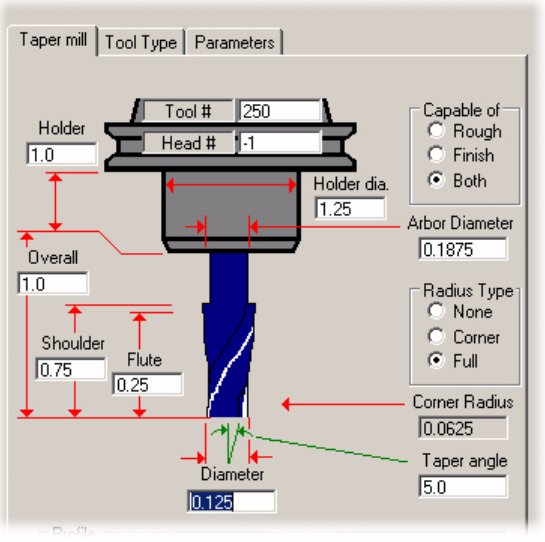
\Documentation\ExampleParts\Electrode.mcx

The sample part includes two toolpaths. The first toolpath is a semi-finish operation over the whole part, while the second finishes the fillet shown in the purple surface:

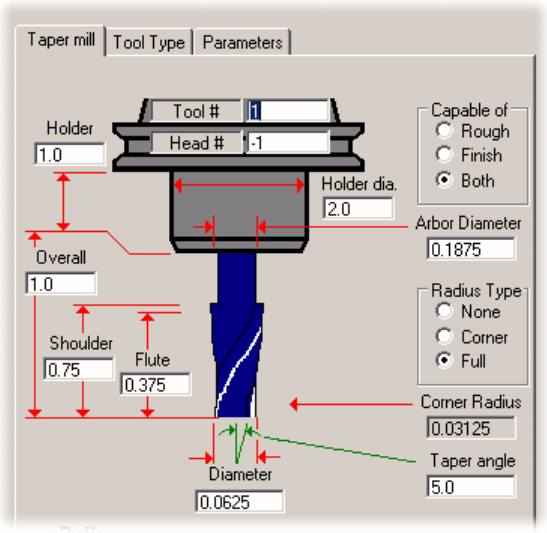


Tooling

For the first toolpath, use a tapered ball cutter with the following dimensions.



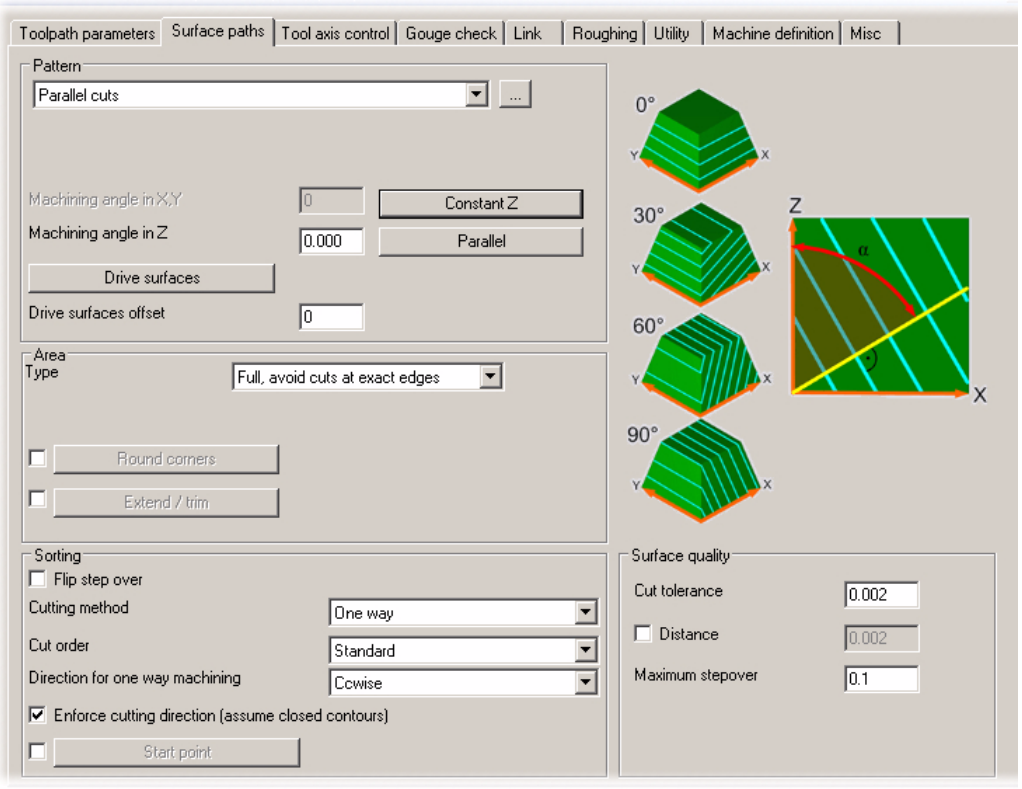
Since the smallest fillet radius in the purple surfaces is 0.05 inches, select a tool with a tip radius very slightly larger than this:



Cutting parameters—semi-finish toolpath

Drive geometry and cutting passes

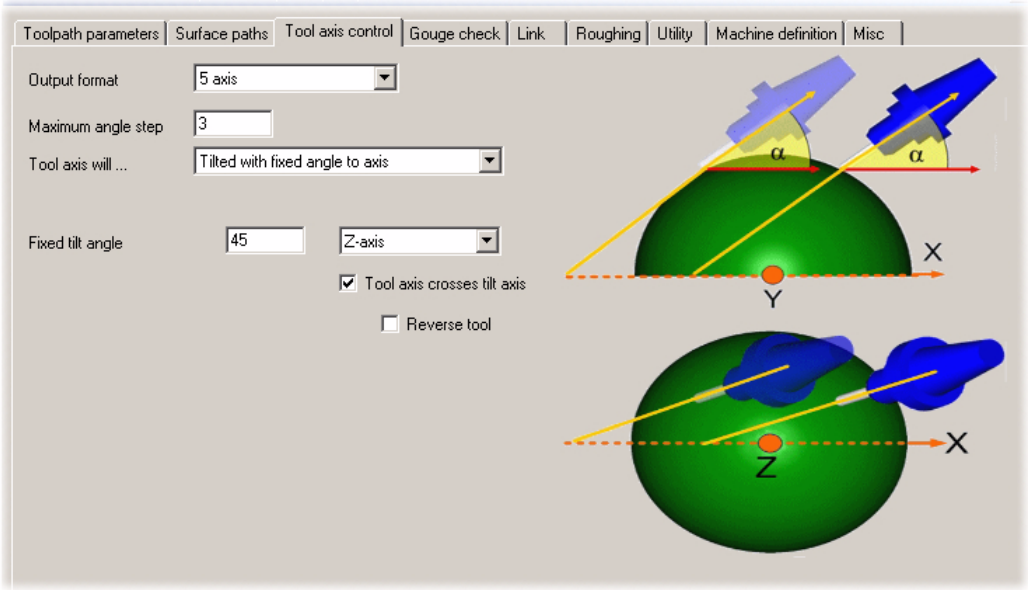
Use the following **Surface paths** settings for the first toolpath:



- From the **Pattern** list, choose **Parallel cuts** and click **Constant Z** to create parallel cuts parallel to the XY plane. Mastercam automatically sets the **Machining angle in Z** to **0.0** degrees, creating horizontal cuts.
- Click **Drive surfaces** and select all of the green and purple surfaces.

Tool axis control and gouge checking

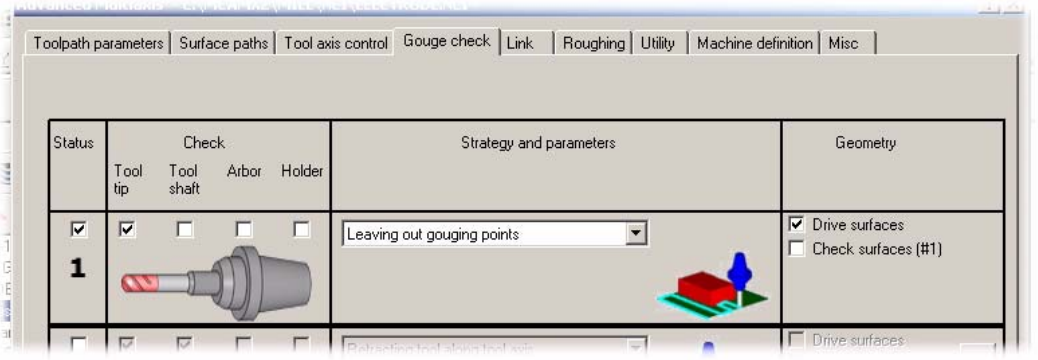
Use the following settings on the **Tool axis control** tab:



This will lock the tool axis to a fixed 45-degree angle to the Z axis throughout the toolpath, resulting in 4+1 axis output.

- Select the **Tool axis crosses tilt axis** option to ensure that the tool axis is pointed directly to the Z axis throughout the toolpath. In other words, when seen from above, the tool will always point to X=0, Y=0. This means that all the Y-axis moves will be substituted by C-axis moves and the only cutting moves will be X- and C-axis moves.

Select the following strategy on the **Gouge check** tab:

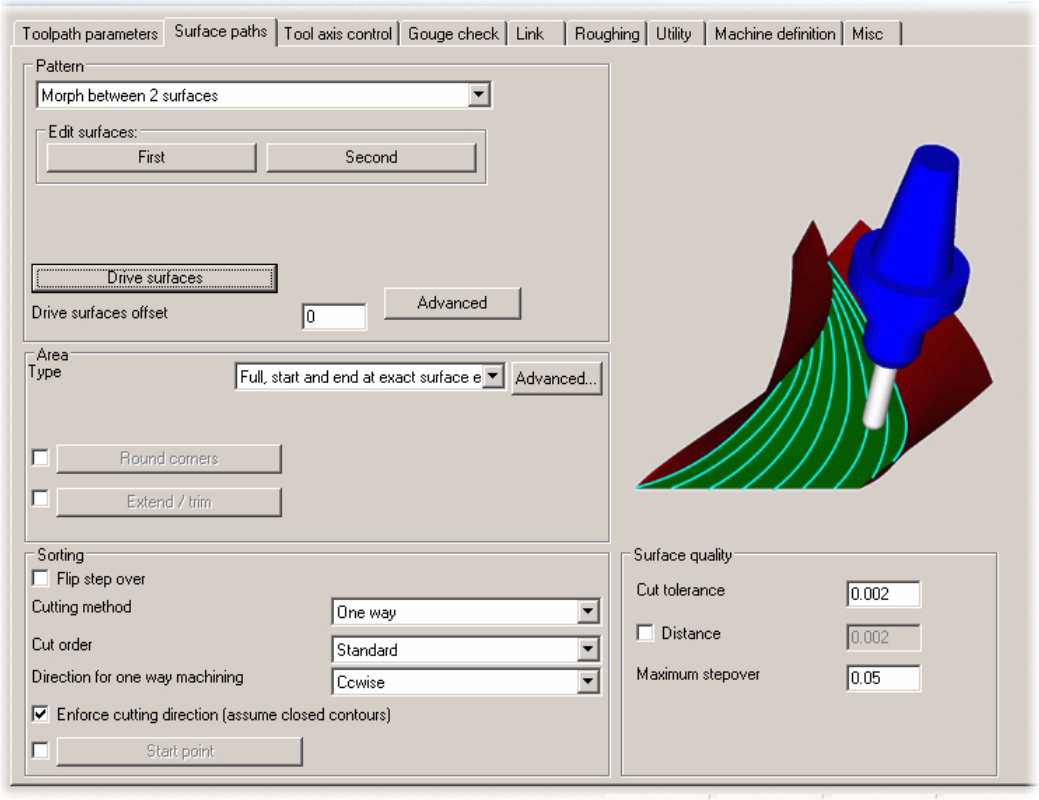


Since this is only a semi-finish operation, you can simply omit any points that are creating gouges. The strategy is only applied to the drive surfaces, not to any check surfaces.

Cutting parameters—finish toolpath

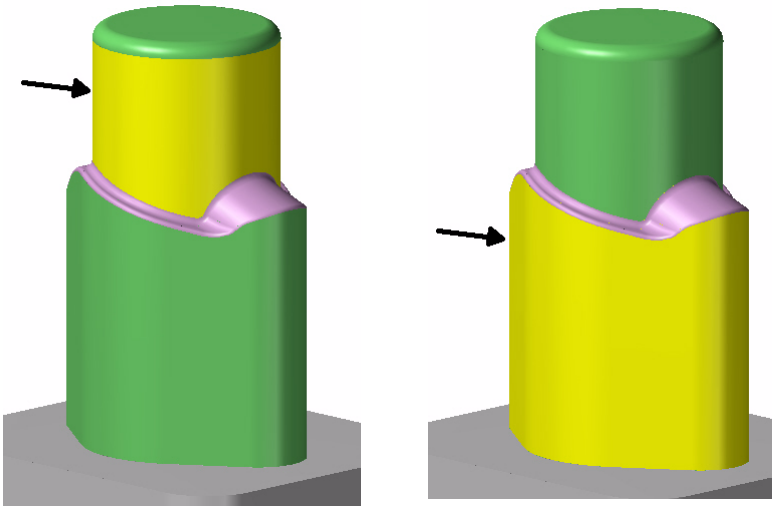
Drive geometry and cutting passes

For the second toolpath, use the following **Surface paths** settings:



- From the **Pattern** list, choose **Morph between two surfaces.** This will create cutting passes whose profile will transition between the boundaries of two surfaces. The profiles will then be projected onto the drive surfaces. Click **First** and select the

upper surface shown on the left. Click **Second** and select the lower surface shown on the right.

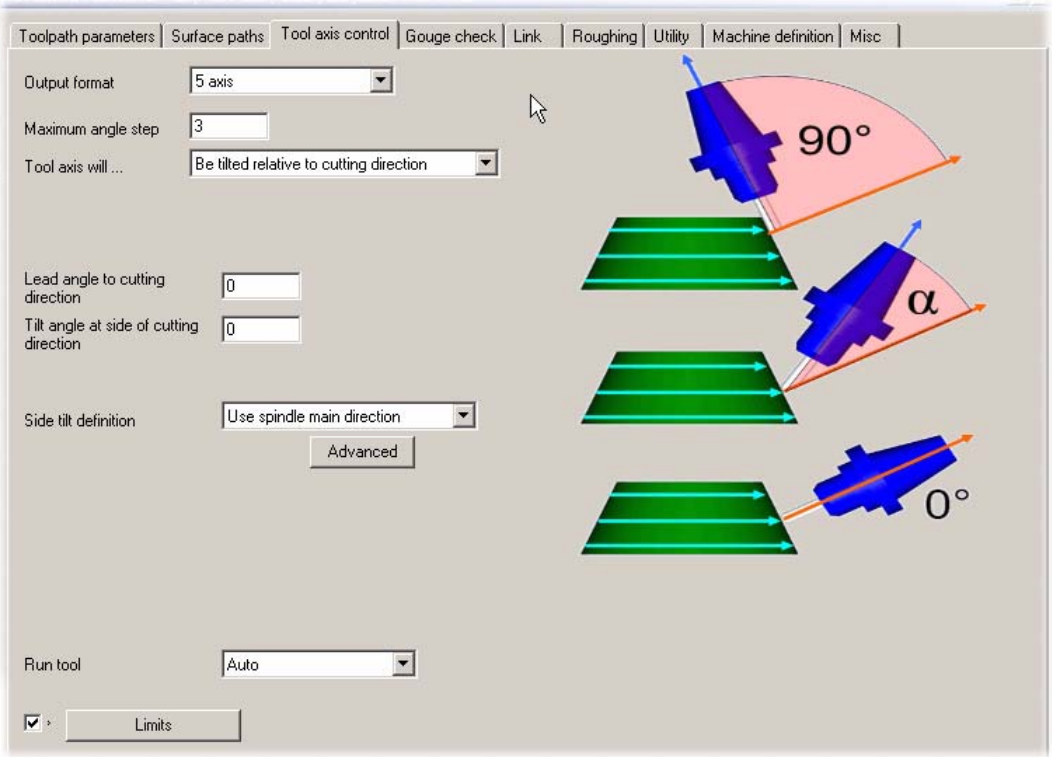


- For **Drive surfaces**, select all the purple surfaces.
- For **Area type**, select **Full, start and end at exact surface edge**. This ensures that Mastercam will create a finish pass along the boundary of the surface. When selecting this strategy, you need to make sure that your surfaces have clean boundaries, otherwise your toolpath will have gaps or chatter. Click **Advanced** to enter a small margin so that the toolpath is not calculated along the absolute boundary:

- Set the **Maximum stepover** to **0.05**. Since the shape of each cutting pass is based on the transition between the two surface boundaries, the spacing between the passes will vary from point to point. Use this value to set the maximum spacing between two passes.

Tool axis control and gouge checking

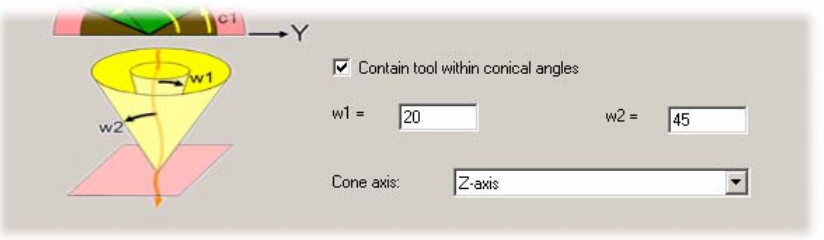
Use the following settings on the **Tool axis control** tab:



- Select the **Be tilted relative to cutting direction** strategy. This will let the tool axis vary more than the previous toolpath, ensuring a finer finish across the entire surface.
- Leave the **Lead angle** and **Tilt angle** at **0**. This will result in the tool axis staying normal to the surface.

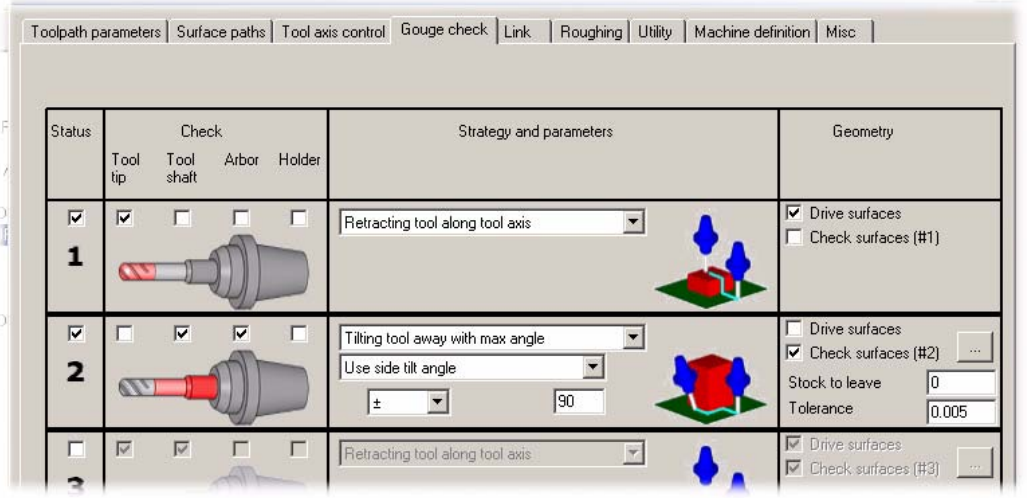
  Typically, when this strategy is selected, you will enter either a lead/lag angle, a side tilt angle, or both. By not entering any tilt angle, this produces the same results as the **Not be tilted and stays normal to the surface** strategy. However, the difference is that later you will select a gouge check strategy that requires a side tilt, so selecting a tool axis strategy that allows a side tilt is necessary.

- Select the **Limits** option and enter the following values:



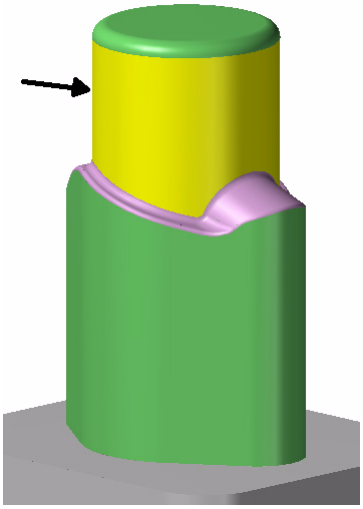
This limits the axis tilt to between 20 and 45 degrees to the Z axis. Instead of locking the tool axis at a 45-degree angle like the previous toolpath, you will allow an extra 25 degrees of movement.

Select the following strategies on the **Gouge check** tab:



- The first strategy is applied to the drive surfaces. In this case, you want to retract the tool along the axis only enough to prevent the gouge, while otherwise continuing the toolpath.
- The second strategy is designed to make sure that the tool shaft and arbor do not gouge the upper part of the electrode when machining the upper fillet. In case of a gouge, you want to simply tilt the tool away from the electrode body.

In the **Check** column, select **Tool shaft** and **Arbor**, and select the **Tilting tool away…** options as shown above. Select the following check surface:

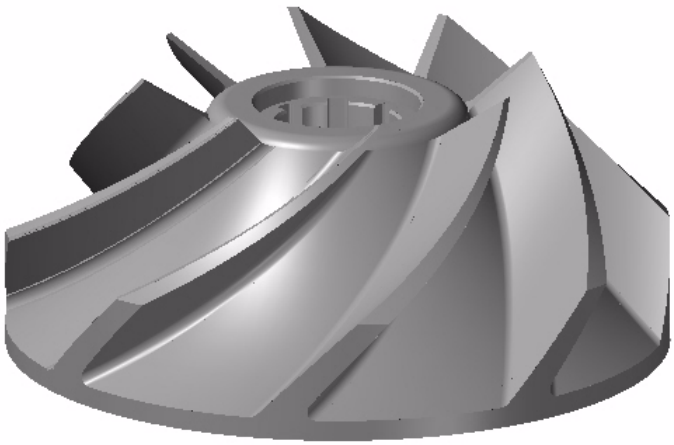


The next several examples all discuss the same part, an impeller. Each example discusses a different operation or type of operation.

Example 5: Impeller Floor Roughing

An impeller is a rotating component of a pump. It transfers energy from the motor that drives the pump to the fluid being pumped by forcing the fluid outwards from the center of rotation. They are used in a variety of demanding applications in which they are subjected to high heat, fluids and steam. Because of this, they are often made from very tough materials. Combined with their difficult shape, this makes them a challenge to machine effectively.

The next several examples all use the same part, the impeller shown here:



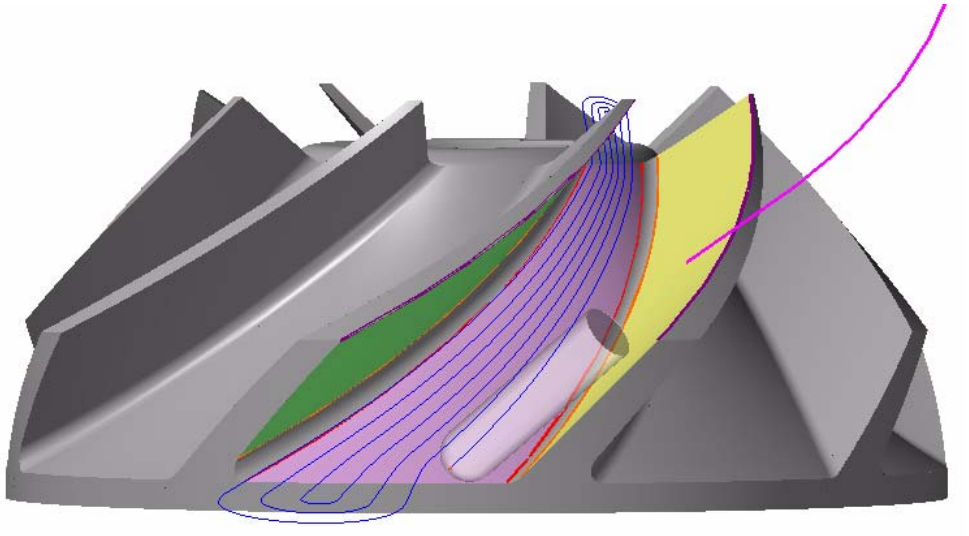
The following machining operations are discussed:

- Roughing between the blades, using both continuous cuts and plunge roughing techniques.
- A semi-finish operation to clean out the undercut area behind the blades.
- Separate finishing operations for the walls and floor.

All of the operations are saved in the following file:

\Documentation\ExampleParts\Impeller.mcx

The roughing strategy discussed in this example creates multiple cutting passes shaped like the following picture:

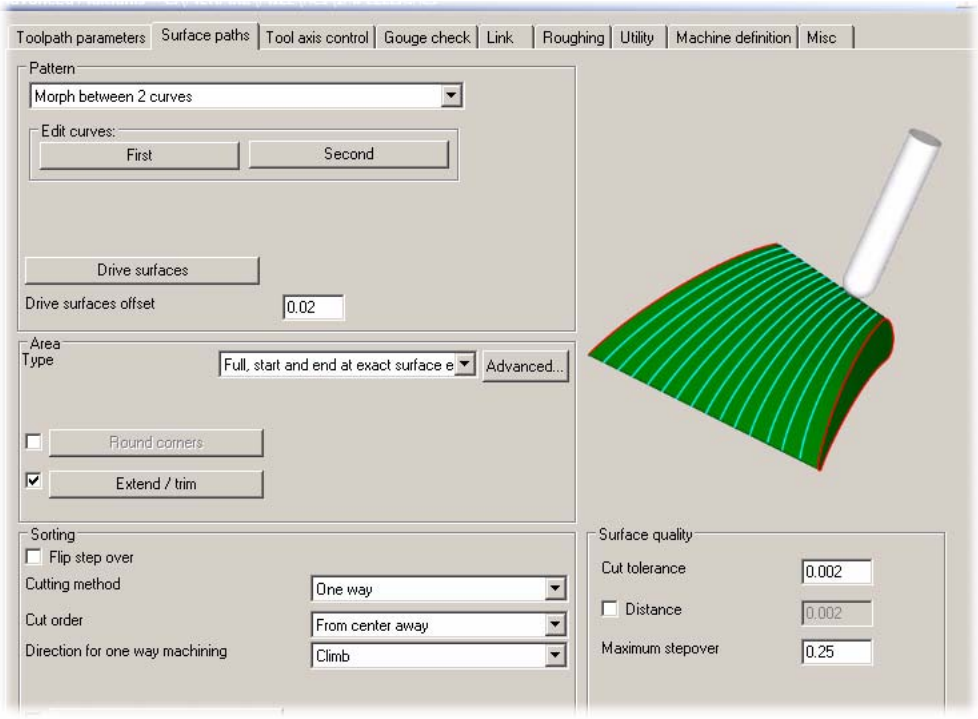


The tool axis is guided by a curve and driven along the floor surface. The tool overlaps the open edges, and is tilted to machine under the front blade.

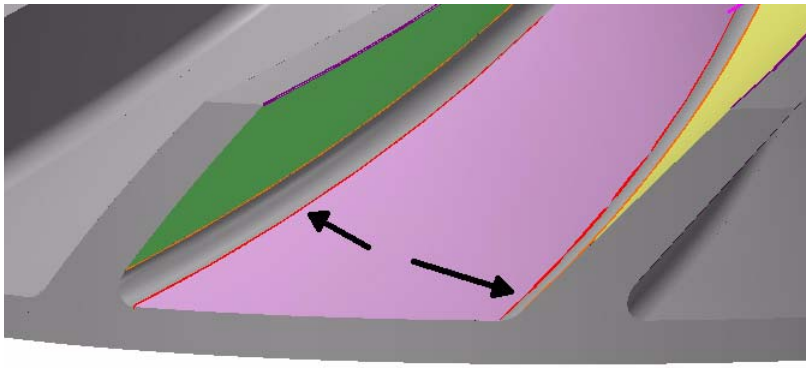
The tool used is a 0.5-inch ball endmill.

Drive geometry and cutting passes

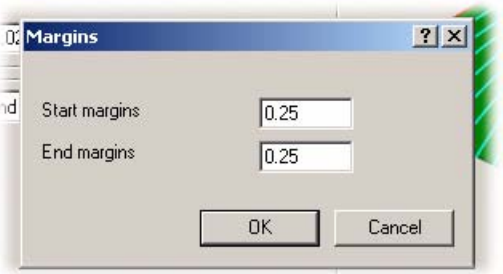
Use the following **Surface paths** settings:



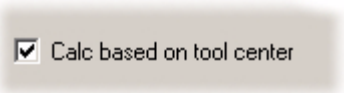
- Select the **Morph between two curves** cutting pattern. This will create cutting passes that transition between two curves and are then projected onto the drive surfaces.
- Select the two drive curves by clicking **First** and **Second** and selecting the two red curves. Select the light purple floor as the **Drive surface**.



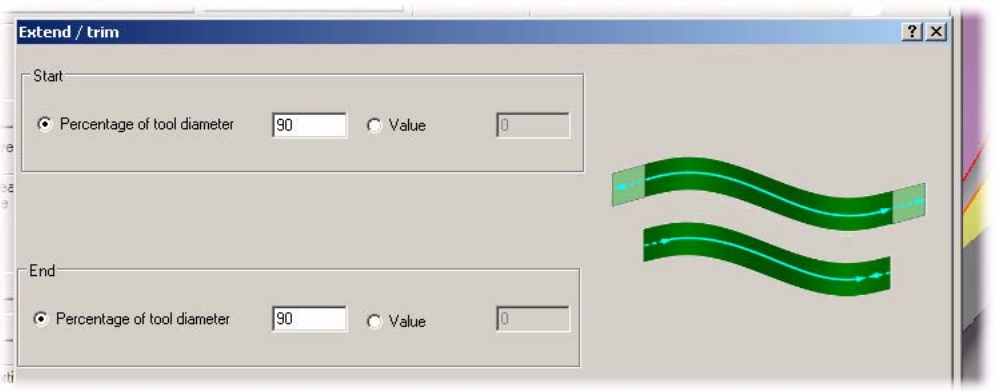
- Enter a **Drive surfaces offset** of **0.02**. This represents the stock that will be left on the drive surfaces for the finishing operation.
- Select the **Full, start and end at exact surface edge** option to force Mastercam to calculate cutting passes at the surface boundary. Select **Advanced** and enter margins of **0.25**—half the tool diameter—so that the edge of the tool is at the boundary, not the center, which would gouge the blades.



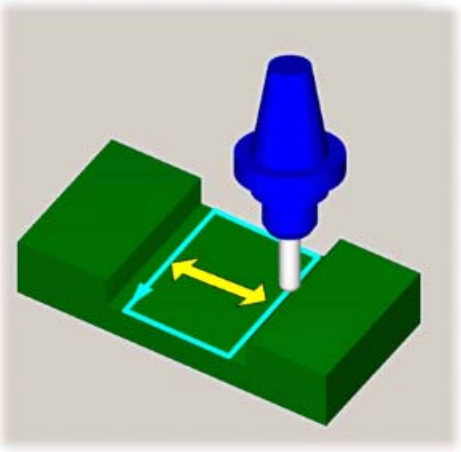
IMPORTANT: This option should be used only with ball nose tools. The following option needs to selected on the Utility tab in order for Mastercam to calculate the toolpath accurately:



- Select **Extend/trim** to extend each cutting pass by 90% of the tool diameter. This forces the tool to cut past the blade boundaries at the open top and bottom, resulting in a clean edge.

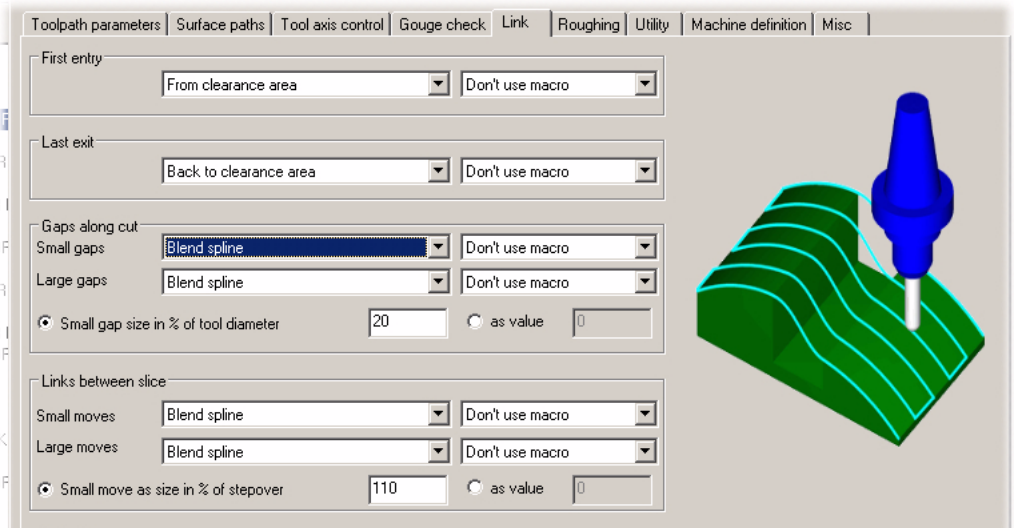


- Select the **From center away** option. Instead of organizing the cuts from one edge to the other, Mastercam will start in the middle and work outwards.



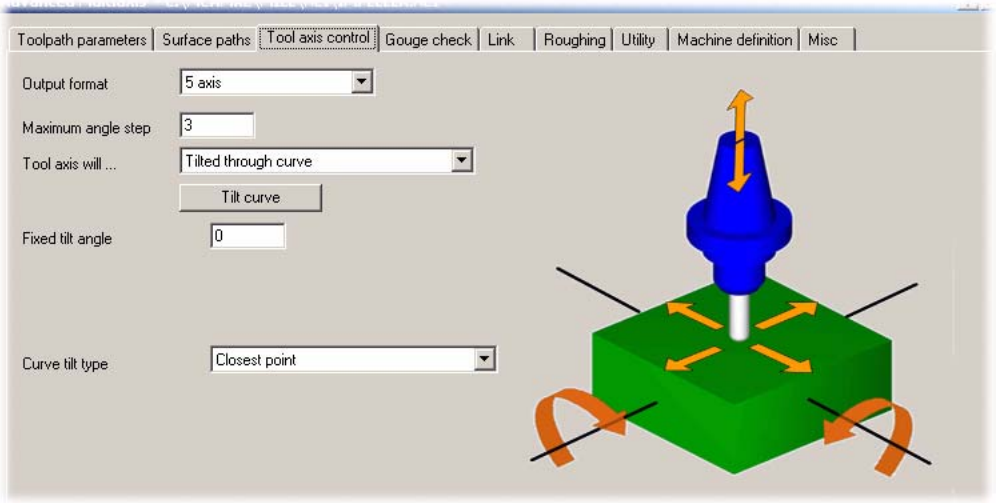
- Enter a **Maximum stepover** of **0.25**, equal to half the tool diameter.

On the **Link** tab, select the **Blend spline** strategy to link the cuts and to control gap motion. This means that whenever Mastercam encounters a gap, it will interpolate a smooth spline from one end of the gap to the other to ensure a smooth transition.



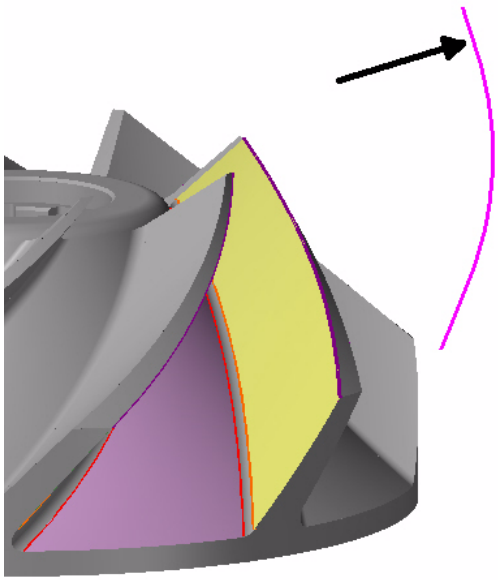
Tool axis control

Use the following settings on the **Tool axis control** tab:



For this toolpath, use a guide curve based on the shape of the impeller blades to control the tool axis.

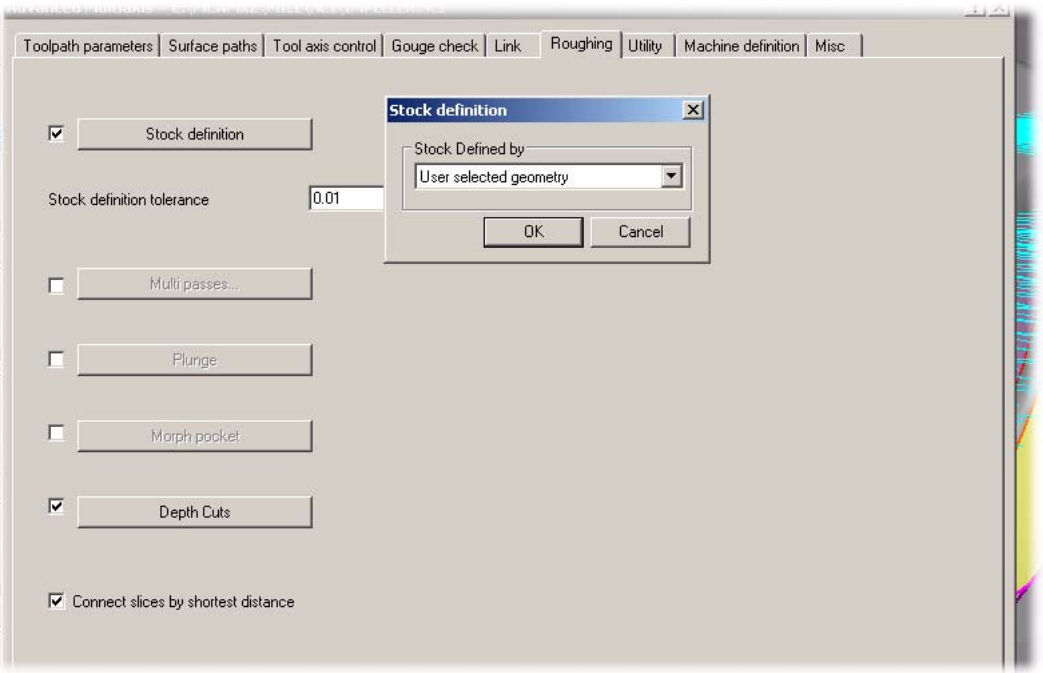
- Select the **Tilted through curve** strategy.
- Select the magenta curve shown below as the **Tilt curve**.



TIP: To create this curve, take one of the boundary curves from the top edge of the impeller blade and rotate it 22.5 degrees. (The eight blades are spaced 45 degrees apart.) Then, translate it above the part in the direction of the surface normal.

Roughing options

Use the **Roughing** tab to define the stock model and configure the roughing passes.



- Select **Stock definition** to apply a stock model. Mastercam will eliminate air cuts outside the stock boundary.
- This part includes stock model geometry. Choose **User selected geometry** and select all the surfaces on level 5.

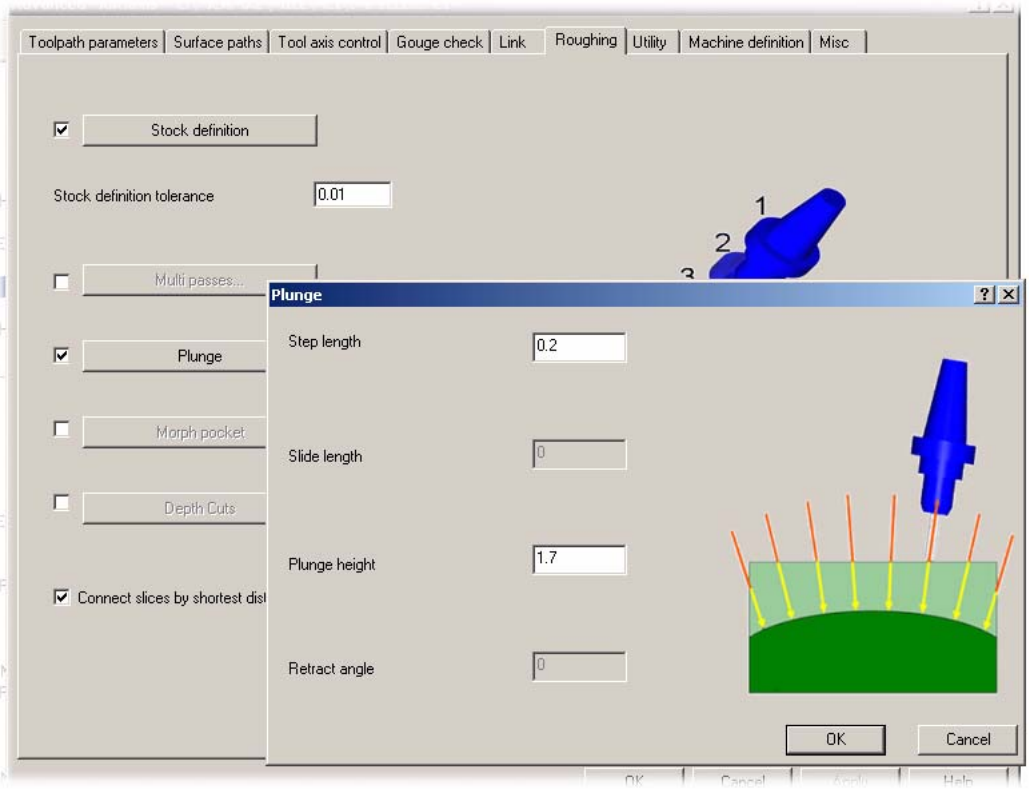
TIP: When you use the **User selected geometry** option, you can select surfaces or solids. To use other types of stock models, such as an external STL file, select **From job setup** and define the stock model in the Machine Group Properties.

- To create multiple cuts, select **Depth cuts** and enter **4** for the number of cutting passes.

Plunge roughing

Because impellers are often made from hard, difficult-to-machine materials, they can be good candidates for plunge roughing. You can accomplish this by making only a few changes to the toolpath. In your **Impeller.mcx** part file, the second operation in the **Floor Roughing** toolpath group demonstrates these changes.

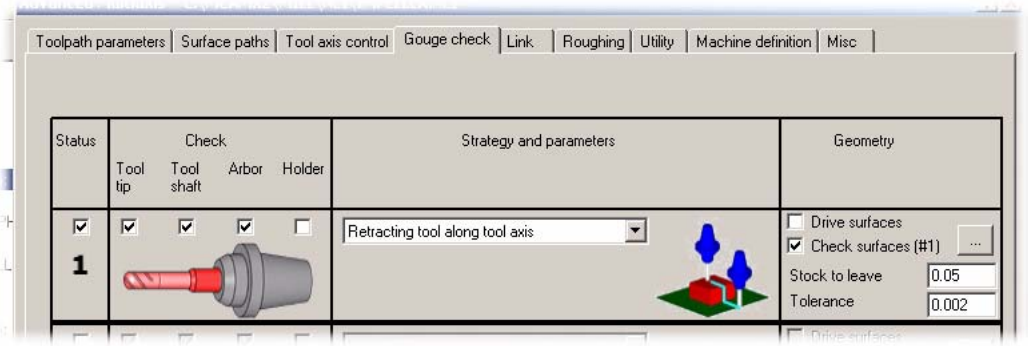
On the **Roughing** tab, de-select **Depth cuts**, and select **Plunge**:



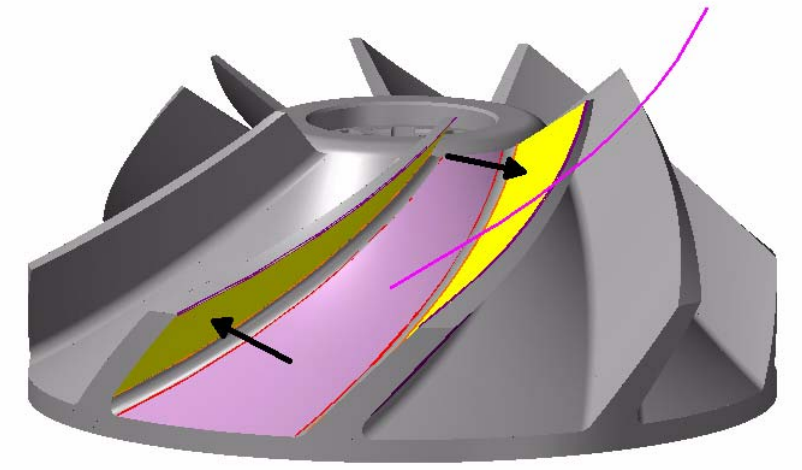
- The plunges will take place along the previously calculated toolpath, spaced according to the **Step length**.

- The **Plunge height** measures the distance from the beginning of the plunge move to the surface. Mastercam automatically trims this if it extends outside the stock boundary.

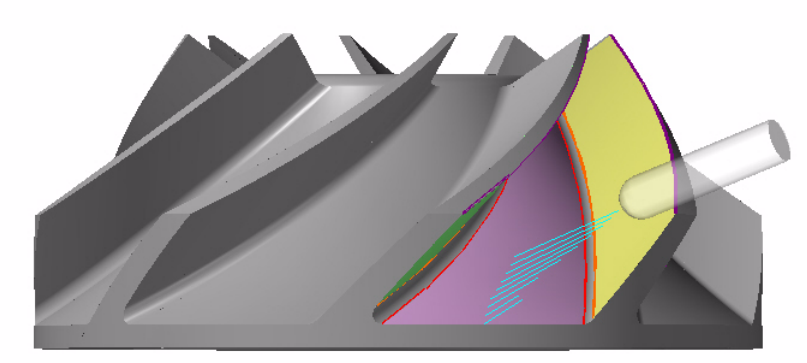
Next, apply a gouge check strategy to make sure the tool doesn't gouge the walls while it plunges.



- Select **Retracting tool along tool axis** strategy so that at each affected location, the tool plunges as far as it can without gouging.
- For **Check surfaces**, select the walls as shown in the following picture. Enter a **Stock to leave** amount of **0.05**.



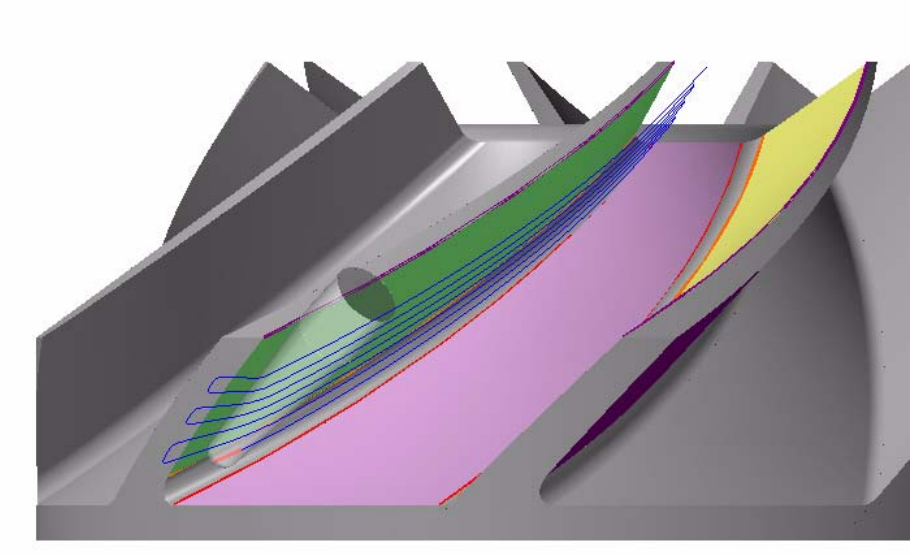
Mastercam will start plunging along the previously calculated toolpath:



Now that you've seen how to rough the floor, continue with the next example to learn how to clean out the back side of the blade.

Example 6: Impeller Blade Semi-finishing

This example continues with the same impeller part used in the previous example. This example discusses the semi-finish operation (shown below) that cleans out the back side of the blade with a tapered tool.

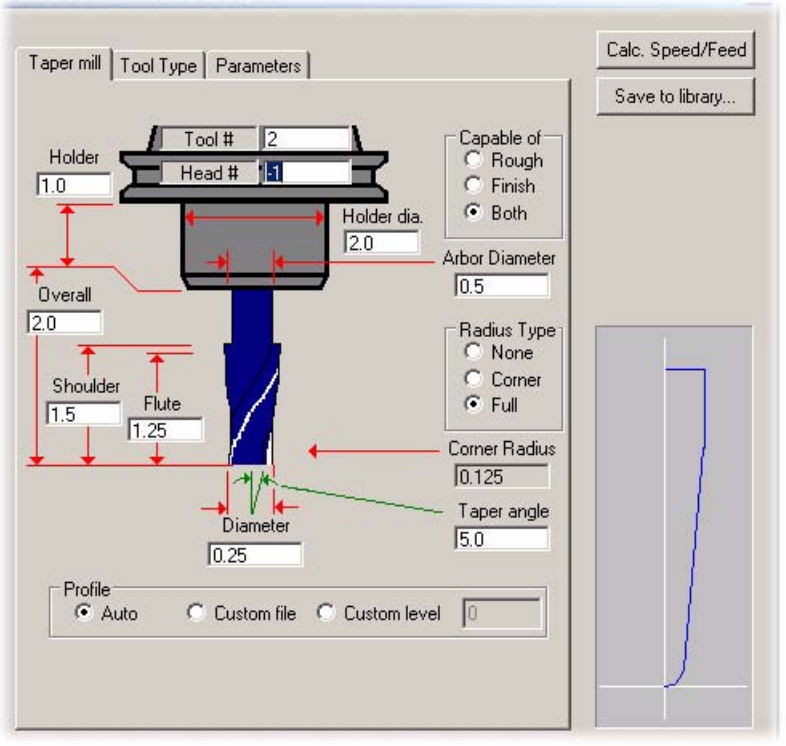


Open the following file:

\Documentation\ExampleParts\Impeller.mcx

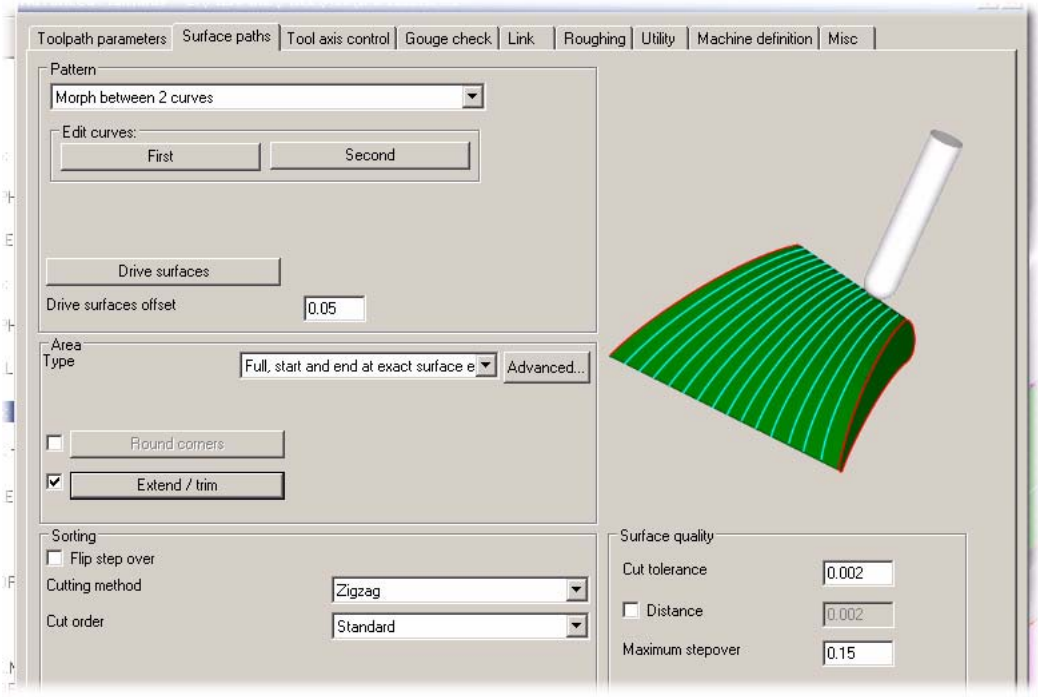
Tooling

Many times there is a small fillet between the blade's back surface and the floor. A tapered ball nose cutter is recommended. Keeping this cutter short and rigid will ensure a quality surface finish.

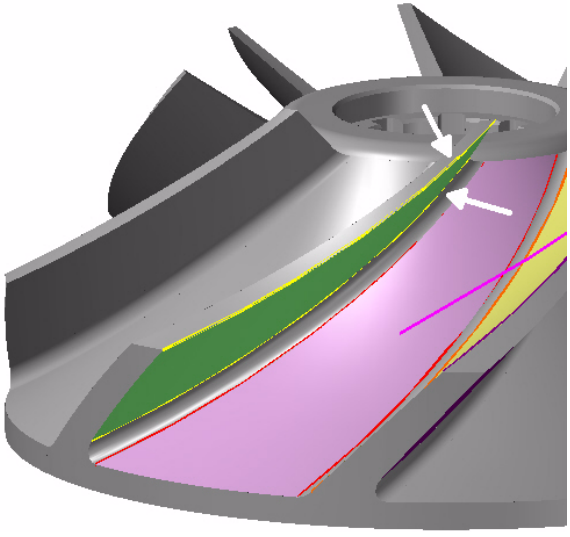


Drive geometry and cutting passes

Use the following **Surface paths** settings:



- Use the **Morph between two curves** cutting pattern. Select the two curves that define the top and bottom edges of the blade.

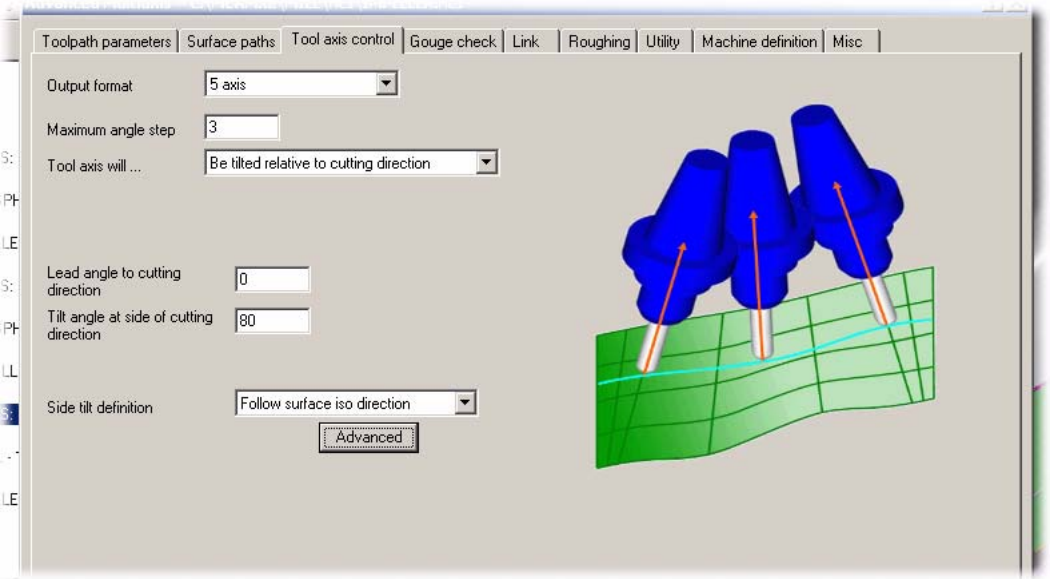


Note: When selecting the curves, the chaining direction does not matter.

- Select the green surface on the back of the blade as the **Drive surface**. Enter **0.05** for the **Drive surfaces offset**, as the stock to leave for the finish operation.
- Define the cutting area as follows:
  - Select the **Full, start and end at exact surface edge** option. Click **Advanced** and enter margins of **0.125** inches, equal to the tool radius.
  - Click **Extend/trim** and extend the passes by **100%** of the tool radius to create a clean edge at the open ends of the part.
  - Choose the **Zigzag** cutting method.

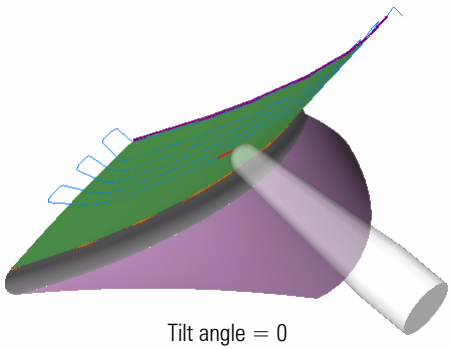
Tool axis control

Use the following **Tool axis control** settings to orient the tool approximately parallel to the surface so that it fits between the blades.

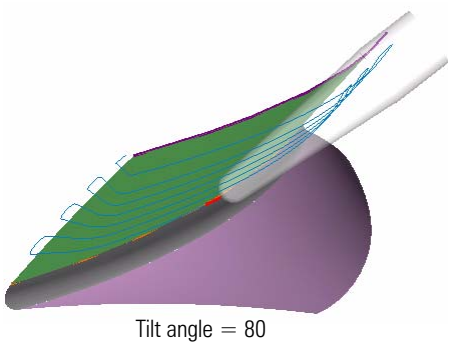


- Select the **Be tilted relative to cutting direction** strategy and enter a **Tilt angle** of **80**.

A tilt angle of 0 (zero) would orient the tool axis approximately normal to the drive surface, as shown in the left picture. Tilting the tool 80 degrees allows it to fit between the blades, and also to remove stock with its side, as shown in the right picture:



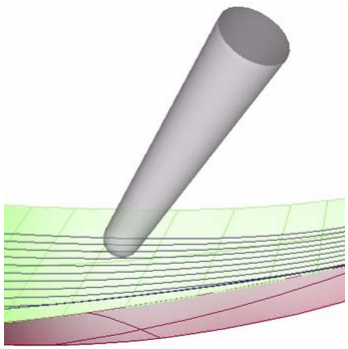
Tilt angle = 0



Tilt angle = 80

- To define the tilt direction, select the **Follow surface iso direction** option. The following pictures explain how this is used.

Every surface has two isometric directions.

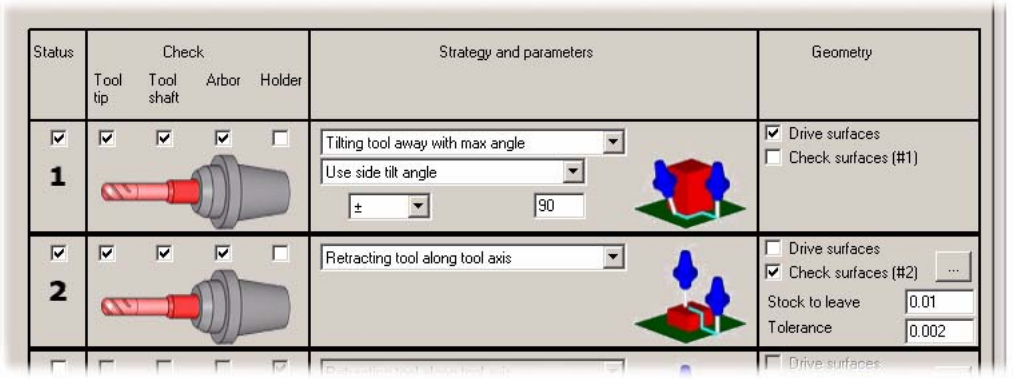


Tilt angle aligned with iso direction.

- If you wish, click the **Advanced** button to enter a fanning distance.

Gouge check strategies

Create two different **Gouge check** strategies, one for the blade and one for the floor:



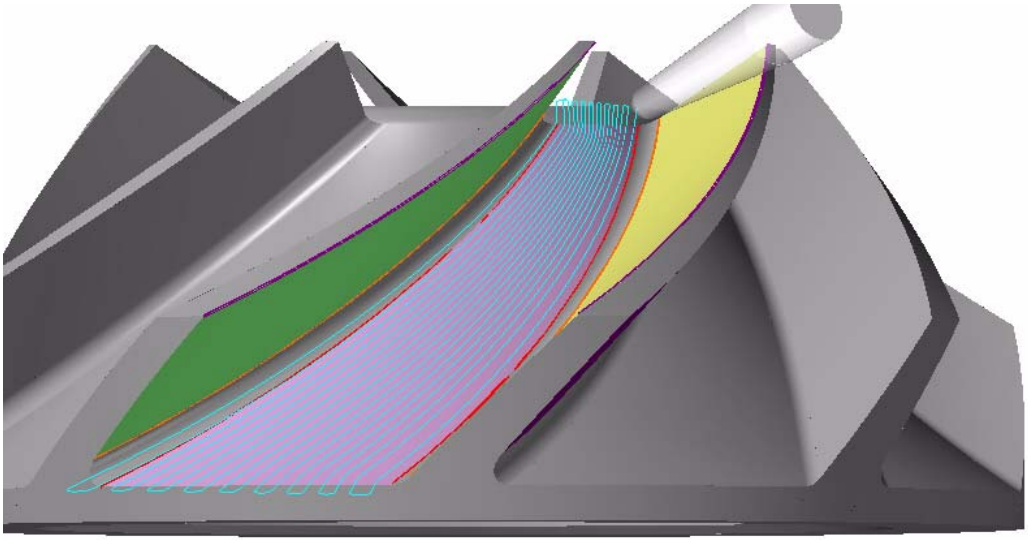
- The first strategy applies to the drive surfaces of the blade. This strategy adjusts the side tilt angle as necessary to avoid a gouge, while still letting you remove stock with the side of the tool.
- The second strategy applies to the floor surface. It simply retracts the tool along its axis enough to prevent gouging the floor, while continuing with the toolpath. Select **Check surfaces** and select the floor surface.

In each case, select the options to check against the **Tool tip**, **Tool shaft**, and **Arbor**. There is no harm if you also select **Holder**, but leaving out unnecessary checks speeds up the toolpath calculation.

The next example continues with this same impeller part. Now that the floor and blades have been roughed, it continues with a finish operation for the floor.

Example 7: Impeller Floor Finishing

This example continues with the same impeller part used in the previous examples. It discusses the toolpath used to finish the floor between the blades.



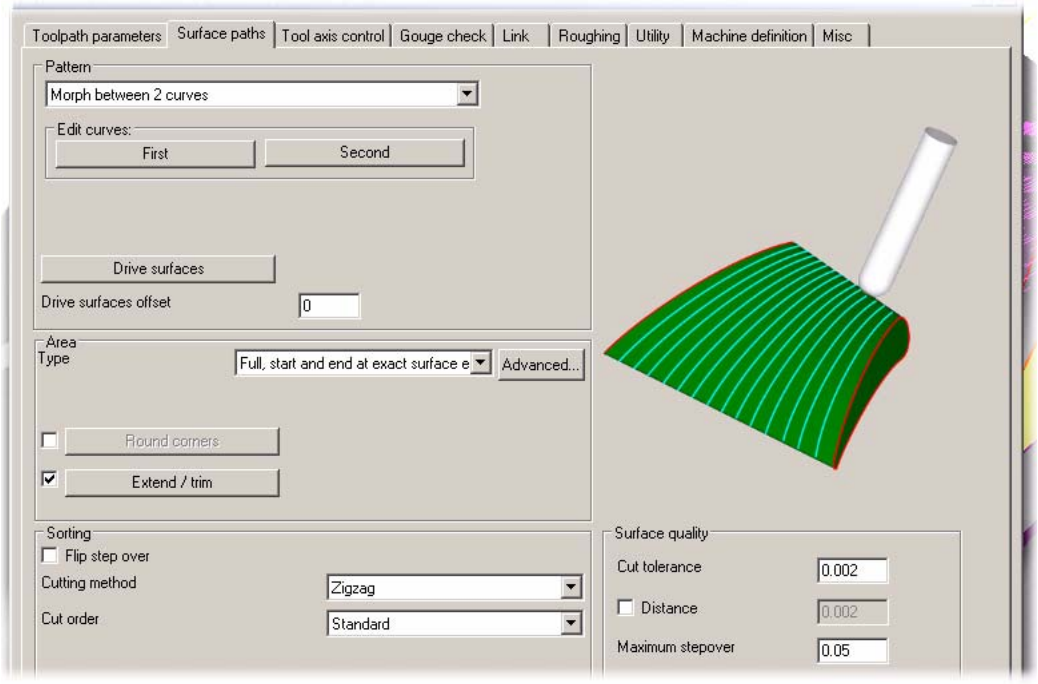
This operation uses the same 0.25-inch tapered endmill used in the previous example.

Open the following file:

\Documentation\ExampleParts\Impeller.mcx

Drive geometry and cutting passes

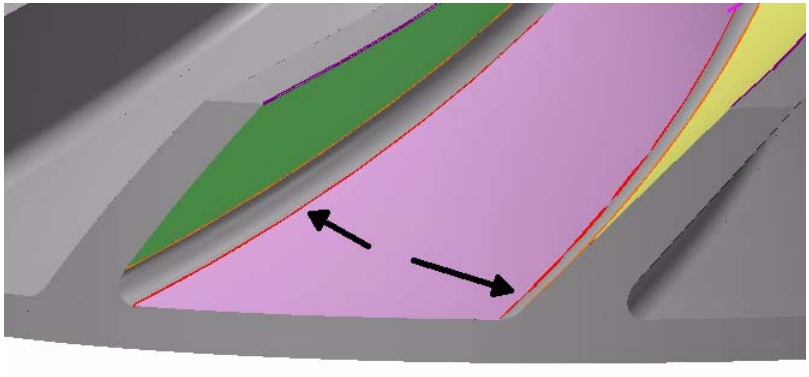
Use the following **Surface paths** settings:



The drive geometry and cut pattern are the same as the floor roughing operation discussed earlier. Many of the other parameters are also discussed in that example.

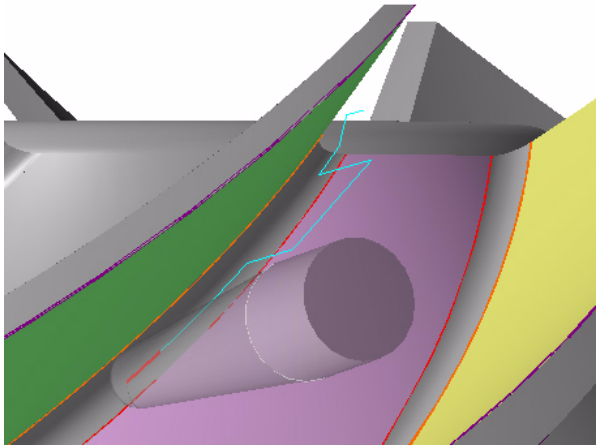
- Select the **Morph between two curves** cutting pattern. This will create cutting passes that transition between two curves and are then projected onto the drive surfaces.

- Select the two drive curves by clicking **First** and **Second** and selecting the two red curves. Select the light purple floor as the **Drive surface**.



- Select the **Full, start and end at exact surface edge** option. Click **Advanced** and enter margins of **0.13** inches.

  Typically, these margins are equal to the tool radius—in this case, 0.125 inches. However, there are some irregularities in the surface boundaries that affect the toolpath:

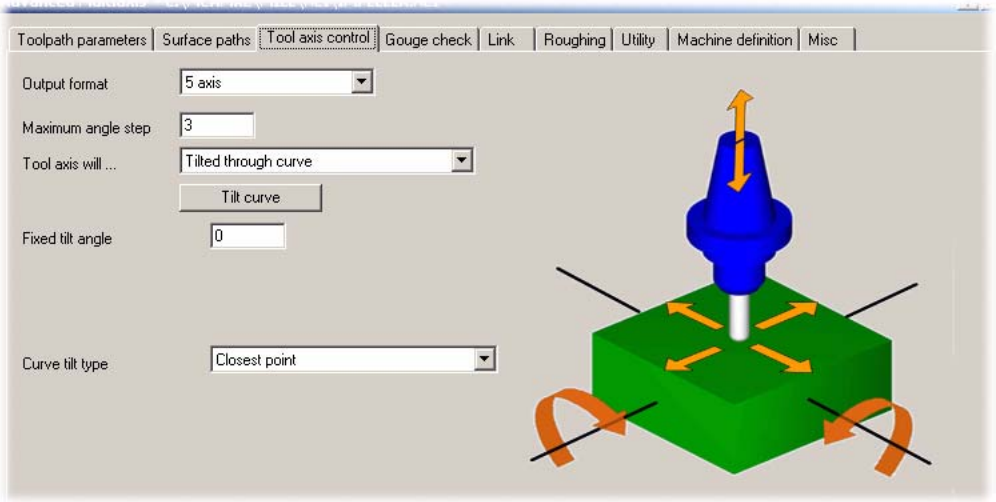


  Adding just a very small additional space cleans up these problems. For this part, the extra margin is not a problem, since the later toolpath that finishes the walls will also run along the same boundary.

- Set the **Maximum stepover** to **0.05** inches.

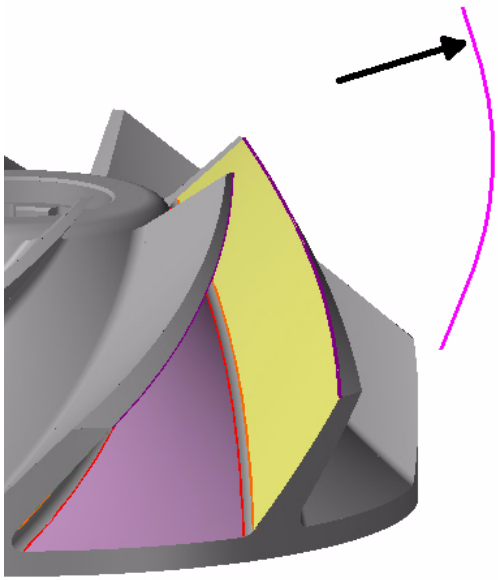
Tool axis control

The **Tool axis control** settings are the same as used for the roughing operation.



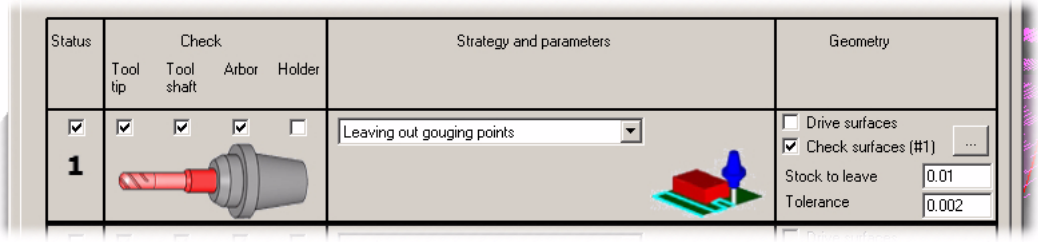
Use a guide curve based on the shape of the impeller blades.

- Select the **Tilted through curve** strategy.
- Select the magenta curve shown below as the **Tilt curve**.



Gouge check strategies

Use the following **Gouge check** strategy:

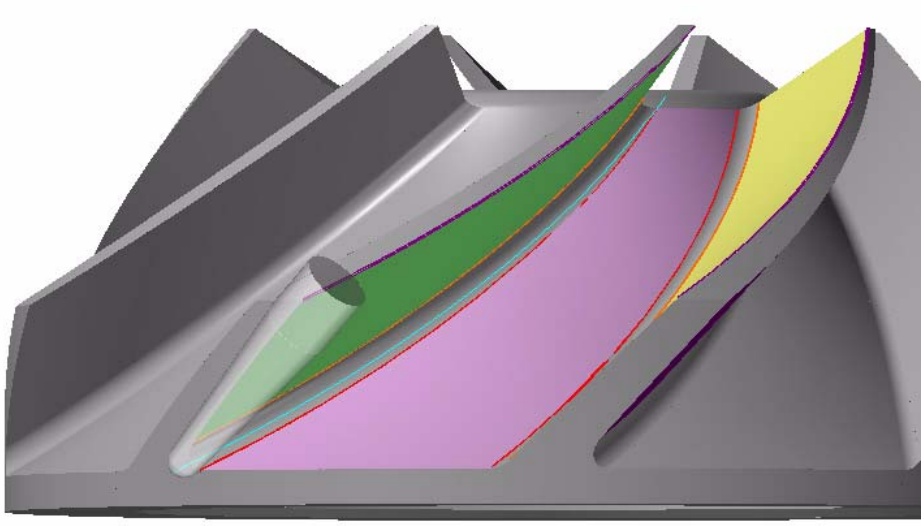


In this case, you are using the two wall surfaces as check surfaces and simply omitting any gouge points from the toolpath, since this is a finish operation and the floor has already been machined once.

The next example illustrates the final operation for the impeller: finishing the blades.

Example 8: Impeller Blade Finishing

This example concludes the impeller machining operations begun in the previous examples. This example discusses two operations that finish each side of the impeller blade. In addition to finishing the blade, the toolpaths cut the fillets between the blade and floor.



Each operation uses the same 0.25-inch tapered endmill as the previous examples.

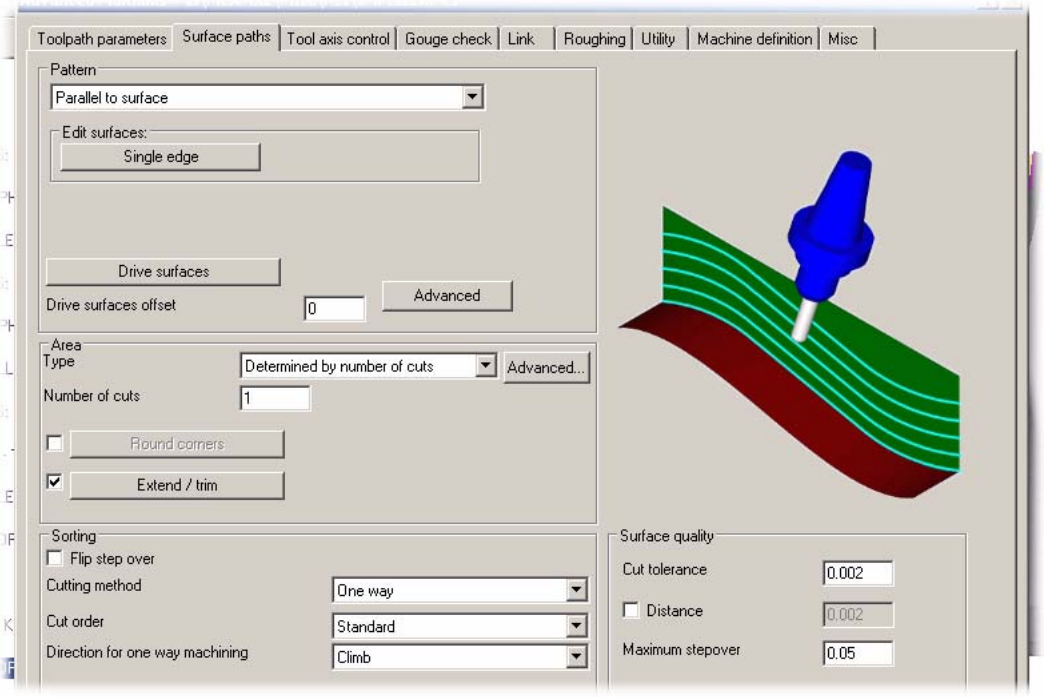
Open the following file:

\Documentation\ExampleParts\Impeller.mcx

There are separate operations for the front and back of the blade. Unless otherwise noted, the toolpath settings are the same for both.

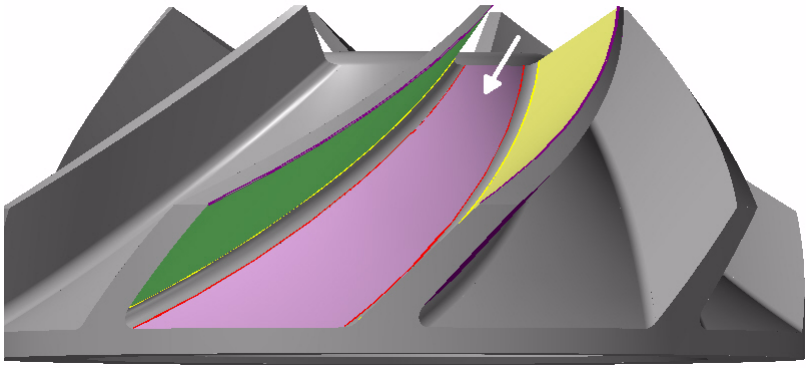
Drive geometry and cutting passes

The goal is to create a single cutting pass along the bottom edge of the blade, finishing the blade with the side of the tool while the ball tip cuts the fillet to the floor surface. Use the following **Surface paths** settings:



- Use the **Parallel to surface** cutting pattern. This creates a series of cutting passes that are each parallel to the boundary of a check surface, projected onto a drive surface.

- Click **Single edge** and select the floor surface as the check surface. This will be the same for each operation.

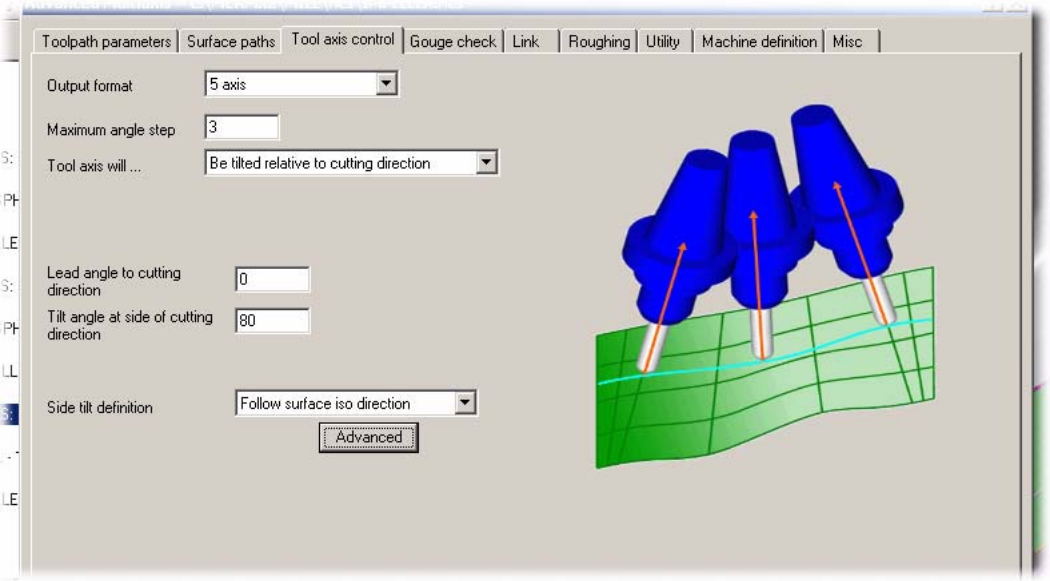


- For **Drive surface**, select one of the blade surfaces—the green surface for one operation, the tan surface for the other.
- The cutting area is defined by the **Determined by number of cuts** option. This tells Mastercam to create a fixed number of cuts no matter how big the drive surface area is. Enter **1** for the **Number** to create a single cut along the check surface boundary.
- Select **Climb** to set the machining direction to climb milling.

Tool axis control

The **Tool axis control** settings are designed to create a swarf cut that finishes the blade surface with the side of the tool. The settings are the same as the semi-finish operation, except that the side tilt angle is

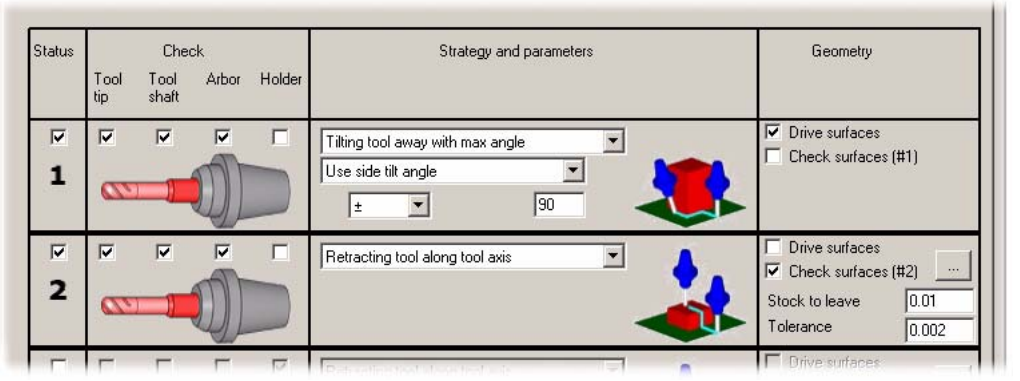
increased to 90 to ensure that the side of the tool maintains contact with the blade surface.



"Tool axis control" on page 104 discusses how the side tilt angle and surface iso directions are used to figure the tool axis angle.

Gouge check strategies

The **Gouge check** strategies—one for the blade and one for the floor—are the same as for the semi-finish operation:



- The first strategy applies to the drive surface of the blade. This strategy adjusts the side tilt angle as necessary to avoid a

gouge, while still letting you remove stock with the side of the tool.

- The second strategy applies to the floor surface. It simply retracts the tool along its axis enough to prevent gouging the floor, while continuing with the toolpath. Select **Check surfaces** and select the floor surface.

Congratulations! You've completed this tutorial and application examples. At this point, you should be able to apply common advanced multiaxis strategies to your own parts. To learn more, click the **Help** button on any tab page to learn more about the options and to see examples of specific settings. You can also consult the *Reference Guide* by selecting it from the **Help** menu.

CNC software, inc.
671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com

Printed in the USA
Mastercam X2 Advanced Multiaxis Tutorial 1-883310-61-X